
DIPLOMA THESIS

Mr. Ing.

Roland Heschl

Numerical Analysis of Rigidity of Welded Sheet Metal Cabinets

Mittweida, 2014

DIPLOMA THESIS

Numerical Analysis of Rigidity of Welded Sheet Metal Cabinets

author:

Mr. Ing.

Roland Heschl

course of studies:

Mechanical Engineering

seminar group:

KM10wMGA-F

first examiner:

Prof. Dr.-Ing. Frank Weidermann

second examiner:

Julien Grell, P. Eng

submission:

Mittweida, July 21, 2014

defense/ evaluation:

Mittweida, 2014

DIPLOMARBEIT

Numerische Analyse von Steifigkeit geschweißter Blechgehäuse

Autor:

Herr Ing.

Roland Heschl

Studiengang:

Maschinenbau

Seminargruppe:

KM10wMGA-F

Erstprüfer:

Prof. Dr.-Ing. Frank Weidermann

Zweitprüfer:

Julien Grell, P. Eng

Einreichung:

Mittweida, 21. Juli 2014

Verteidigung/Bewertung:

Mittweida, 2014

Bibliography:

Heschl, Roland:

Numerical Analysis of Rigidity of Welded Sheet Metal Cabinets. - 2014. - VI, 59 S.

Mittweida, Hochschule Mittweida, Faculty of Mechanical Engineering,

Diploma Thesis, 2014

Abstract:

The aim of the present paper is to define the most efficient way of using the FEM analysis in the design process of welded sheet metal cabinets in the application of casino gaming machines with particular focus on the optimization of cost and rigidity.

In the course of this, representative and potentially revealing load cases and constraint sets that represent worst case real world scenarios shall be defined.

Furthermore, simplification and idealization capabilities of the FEA software (Creo Simulate 2.0) and potential advantages of interworking with the integrated CAD software (Creo Parametric 2.0) will be explored and outlined.

The outcome of the investigations carried out within that thesis provide a detailed explanation of the most efficient analysis procedure as well as simulated magnitudes of deformation of current cabinets resulting from specified loads acting as a benchmark and a guide value for the design of new products.

Contents

Contents	I
List of Figures	III
List of Abbreviations.....	VII
1 Introduction	9
1.1 <i>Background.....</i>	9
1.2 <i>Purpose.....</i>	10
1.3 <i>Objectives</i>	11
1.4 <i>Creo Simulate</i>	12
2 Constraints and Load Cases for Cabinets.....	13
2.1 <i>Side Force on an Open Door.....</i>	14
2.1.1 <i>Defining the Load</i>	14
2.1.2 <i>Constraints for a Freestanding Cabinet</i>	15
2.1.3 <i>Constraints for a Cabinet Mounted to a Base</i>	19
2.1.4 <i>Reinforcement with MDF Base Plate</i>	22
2.2 <i>Rigidity of a Base Plate of a Free Standing Cabinet</i>	23
3 Method and Theory	25
3.1 <i>Preprocessing.....</i>	25
3.1.1 <i>Simplified Representation.....</i>	25
3.1.2 <i>General Settings</i>	26
3.1.3 <i>Material</i>	27
3.1.4 <i>Idealization</i>	28
3.1.5 <i>Connections</i>	29
3.1.6 <i>Mesh</i>	32
3.1.7 <i>Constraints.....</i>	35
3.1.8 <i>Loads</i>	38
3.2 <i>Executing the Simulation.....</i>	39
3.2.1 <i>Static Analysis.....</i>	39
3.2.2 <i>Convergence.....</i>	41
3.3 <i>Post Processing</i>	45
4 Results	49

4.1	<i>Limitations.....</i>	49
4.2	<i>Evaluation</i>	49
4.3	<i>Results of Existing Cabinets.....</i>	50
4.3.1	Oxygen Slant Top Cabinet	50
4.3.2	Axxis Cabinet	51
4.3.3	Oxygen Upright Cabinet.....	52
4.3.4	Stenmark Cabinet	52
5	Conclusion	55
5.1	<i>Achievement</i>	55
5.2	<i>Further Applications</i>	56
	Bibliography.....	57
	Statement of Authorship	59

List of Figures

Figure 1-1: Products.....	9
Figure 2-2: Force on open door	14
Figure 2-3: Force acting on mounting points	15
Figure 2-4: Preliminary constraints for freestanding cabinet	16
Figure 2-5: Results with different places of constraints	17
Figure 2-6: Final constraints for freestanding cabinet	17
Figure 2-7: Results with more restrictive constraints.....	18
Figure 2-8: Preliminary constraints for a cabinet mounted to a base	19
Figure 2-9: Constraints for base support	20
Figure 2-10: Final constraints for a cabinet mounted to a base	21
Figure 2-11: Connections for MDF base plate	22
Figure 2-12: Testing force for base plate	23
Figure 3-13: Removal of redundant geometry features.....	25
Figure 3-14: Default Interface	26
Figure 3-15: Material Assignment.....	27
Figure 3-16: Material Definition for STEEL and MDF	27
Figure 3-17: Detect Shell Pairs.....	28
Figure 3-18: Review Geometry.....	28
Figure 3-19: Shell pairs derived from sheet metal	29
Figure 3-20: Weld geometry type	30

Figure 3-21: Weld feature adoption from CAD model	30
Figure 3-22: Connections for MDF board	31
Figure 3-23: Weld connections derived from CAD model	31
Figure 3-24: Common problem of weld lines in small holes.....	32
Figure 3-25: Mesh with h- elements (left) and p- elements (right) [ToRo2012]	33
Figure 3-26: Midsurfaces only	33
Figure 3-27: AutoGEM settings	34
Figure 3-28: Create and validate mesh	34
Figure 3-29: Mesh.....	35
Figure 3-30: Rigid Link for cabinet leg.....	36
Figure 3-31: Displacement constraint for freestanding cabinet.....	37
Figure 3-32: Fixed mounting bolt holes	37
Figure 3-33: Point constraints for base support.....	38
Figure 3-34: Load on virtual point.....	39
Figure 3-35: Static analysis	40
Figure 3-36: Settings for static analysis.....	40
Figure 3-37: Diagnostics for analysis	41
Figure 3-38: Convergence with h- elements (top) and p- elements (bottom) [ToRo2012]	42
Figure 3-39: Definition of an angle measure.....	43
Figure 3-40: Setup for convergence criteria	44
Figure 3-41: Results of convergence passes of different methods	44
Figure 3-42: Comparison calculation time of convergence methods.....	45

Figure 3-43: Result windows for rotation	46
Figure 3-44: Result windows for displacement	46
Figure 3-45: Settings for result windows – displacement and rotation	47
Figure 3-46: Settings for result windows – stress	47
Figure 4-47: Top rotation – Oxygen Slant Top.....	50
Figure 4-48: Leg displacement – Oxygen Slant Top.....	51
Figure 4-49: Top rotation – Axxis.....	51
Figure 4-50: Top rotation – Oxygen.....	52
Figure 4-51: Top rotation – Stenmark.....	53
Figure 4-52: Leg displacement – Stenmark	53

List of Abbreviations

FEA	finite element analysis
FEM	finite element method
AWP	amusement with prices
CAD	computer aided design
MDF	medium density fiber
SPR	software performance report
RAM	random access memory
CPU	central processing unit
PC	personal computer
QC	quick check
SPA	single pass adaptive
MPA	multi pass adaptive
RMS	root mean square

1 Introduction

This diploma thesis deals with defining load cases and proper procedures for the numerical analysis of rigidity of gaming machine cabinets. The following section contains a brief presentation of the company GTECH S.p.A. and its products as well as the purpose and objectives of the thesis.

1.1 Background

GTECH S.p.A. is an Italian joint stock company operating in many different areas of the international gaming industry with a global workforce of about 8500 employees. Under its brand name SPIELO the company develops and produces slot machines for various markets like casinos, video lottery and AWP (amusement with prizes). The gaming machines industry employs about 900 people and produces about 35.000 gaming machines per year. The main development facilities are located in Moncton, Canada and in Graz, Austria. See Figure 1-1 for examples of currently available products in the cabinet segment.



Figure 1-1: Products

1.2 Purpose

At present, the design of welded sheet metal cabinets is based on experience and assessment of existing designs and the validation of physical prototypes. The established approach of designing cabinets was working quite well, when requirements of new products in regards to features, aesthetics and serviceability were less complex. But recent developments create increasing challenges in incorporating all requirements, keeping the cost low and providing enough rigidity at the same time.

Varying cabinet designs make it more difficult to adopt proven designs or to benefit from experience of previous products. New designs often use more displays with bigger sizes and fundamentally different integration, big add-ons on top of the cabinet and new ways of door opening with complicated hinging mechanisms. As a consequence, doors are constantly getting bigger and heavier and the cabinets have to support them and have to be designed with accordingly. Furthermore, lots of new developments and technologies are coming up, which will force cabinet designs to become even more diverse than before.

In addition to the consistent challenge of designing housings that are able to support heavier doors, the inside of the cabinets become smaller with an expanding requirement to house even more and bigger components. Electronic platforms will need more room to incorporate big 3D graphic cards, and in some cases even additional logics are necessary. Moreover there is an increasing demand of integrating many different peripheral devices like bill validators, ticket printers and coin acceptors that are requested by certain customers or markets. Also many different market requirements need to be incorporated on an optional basis.

At the same time it is important to keep the serviceability at least on the same level as predecessor products. The unavoidable consequence of this development is that it is getting harder and harder to provide enough structure in a new cabinet design, due to increased space constraints.

Recently developed products from GTECH had to run through several prototyping loops until they were mature enough for serial production. In the course of this process, structural issues were playing a big role which led to the idea and desire to investigate in research and tests, whether structural analysis with numerical FEM simulation is able to improve the development process in an effective way and is able to save cost for prototyping and design loops.

1.3 Objectives

Mechanical design for all products of the company GTECH is modeled in the 3D CAD software Creo Parametric 2.0 (former Pro/ENGINEER) from company PTC. In this software, sheet metal is designed with the help of a special module that assists in creating bending rounds and other specific features of sheet metal. Different types of weld lines are already created as a 3D feature in the CAD model in order to connect all individual sheet metal parts of a cabinet unit. This is basically done to support the generation of the 2D drawing. But it would be a great benefit if all these definitions can be adopted by an FEM analysis as well.

In order to utilize the definitions already available from the regular design work, the company GTECH invested in a license of the FEM module Creo Simulate 2.0 (former MECHANICA). This module is fully integrated with Creo and allows benefiting from already defined sheet metal models and welding connections and makes it possible to modify the design more quickly by allowing switching between the different modules easily.

FEM simulation is a time consuming operation and the number of elements in the FEM mesh will mainly determine the required computation time. Thus, it is one of the main goals, when setting up the simulation, to reduce the number of elements in the mesh without affecting the accuracy of the result. Sheet metal structures are very big compared to their thickness. When creating a mesh with regular solid elements, a lot of elements need to be created. Therefore thin sheet metal parts are best analyzed with flat shell models rather than solid elements. Shell elements assume the sheets are infinite thin, but have the same properties as solid plates with a certain thickness and material. This reduces the number of elements drastically and still gives a reasonable result with far shorter calculation time. Creo Simulate is able to create these shell elements automatically from sheet metal models of Creo Parametric. This outlines another benefit of the integrated module.

In the following chapters, this thesis will elaborate the abilities of the Creo FEA module in analyzing rigidity of sheet metal weld assemblies in the application of gaming machine cabinets. The aim of this paper is to find and define an effective way to use FEM analysis in existing design processes. Therefore, an efficient way of simplifying the geometry of parts should be found and shell elements as well as weld features should be utilized and adopted from the CAD model.

Required simulation settings and procedures should be worked out to determine the fastest and most convenient way to get a result that provides sufficient and confident accuracy. Furthermore, proper and representative constraining sets should be found for the different types of cabinets and common load cases should be defined, which can act as criteria of comparison for new products.

1.4 Creo Simulate

Creo Simulate is an analysis tool conducting finite element analyses (FEA). It uses the finite element method (FEM) to model and solve complex structural, fluid or multi-physics problems. In the mathematical modelling of a lot of physical phenomena partial differential equations occur that cannot be solved analytically. Therefore the finite element method provides a powerful technique to numerically approximate a solution that will be close enough to the unknown analytical result.

By splitting up the whole into a finite number of small elements with simpler shape, more but therefore solvable equations can be obtained and calculated with the help of a computer. The accuracy of that method is basically determined by the number and type of elements created.

The general result of a FEA simulation is presented by the displacement of all nodal points of the elements. Based on that, strains, stresses, rotations and reaction forces will be calculated.

With the help of Creo Simulate 2.0 without the advanced simulation extension the following analysis can be performed:

- Linear statics - including contact analysis
- Natural frequencies with modal analysis
- Factor of safety against buckling
- Stationary thermal analysis - with conduction and radiation

The following analysis can only be done with Creo Advanced Simulation Extension:

- Static and dynamic with nonlinear and anisotropic material behavior
- Large displacement
- Transient stationary thermal analysis
- A few advanced types of connections and idealizations

The subsequent analysis takes only linear type of material behavior into account as permanent deformation is not of interest in this thesis. It is assumed and ensured that stress levels stay below yield strength.

2 Constraints and Load Cases for Cabinets

This chapter deals with definition of representative and potentially revealing load scenarios and cases on gaming machine cabinets that should act as benchmark or guide value. These load cases should be analyzed while designing new products and should be compared to results from previous designs that have proven to have sufficient stiffness properties.

A gaming machine cabinet doesn't have to fulfill formal external or internal requirements regarding maximum displacement or deformation due to a certain load. Neither does it usually have to bear high loads as a matter of its main function.

Anyhow, it has to withstand certain external loads that will arise during handling and will affect the sub functions of the machine like door locking and latching. Gap measures and alignment and therefore the perception of quality of the whole product will be affected as well. Therefore, load and constraint cases that are typical and critical to gaming machines will be defined and explained in the following sections.

The resulting figures of displacement or rotation of existing products, calculated for these cases, will be shown in chapter 4.3, and should provide guidance for numerically calculated results of future designs.

2.1 Side Force on an Open Door

2.1.1 Defining the Load

One of the most severe loads on a cabinet is presented by somebody pushing or pulling on open doors. This could be a top lifting door, which is pushed on the side or a regular, side-hinged door that is pushed or pulled in vertical direction. See Figure 2-2.

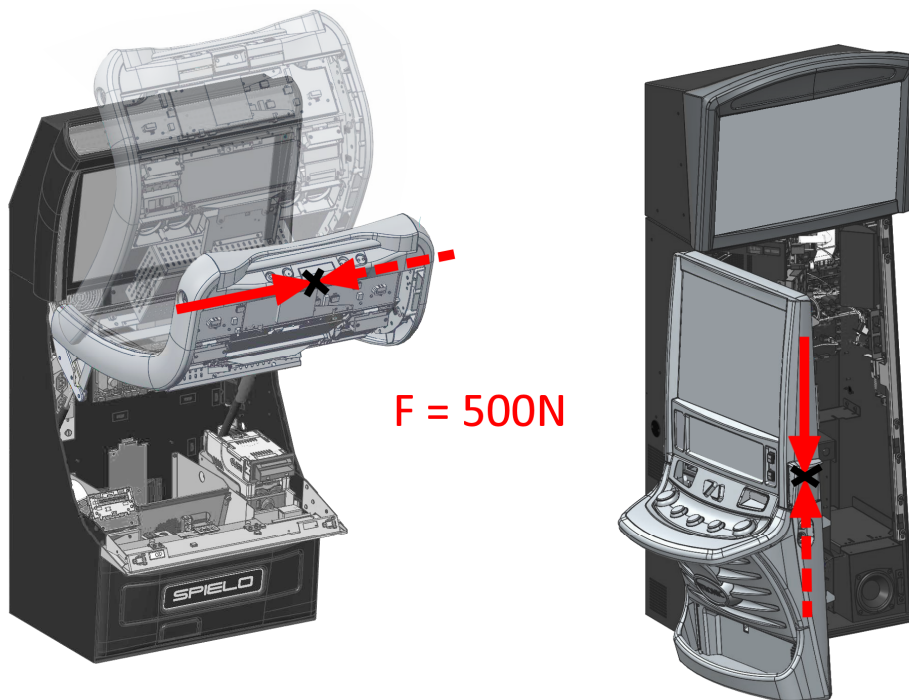


Figure 2-2: Force on open door

To simulate a load scenario like this a force of 500N is applied to the center of the virtual door and as far away from the back of the cabinet as the most forward point of the door is located. Therefore the force is applied to a point on the defined location and is set to act on the mounting spots of the cabinet that bear the door (See Figure 2-3).

The simulation software applies the appropriate load to the mounting points as if it were applied directly there. In doing so the leveraging effect is considered and it is simulated as if the door is not adding any structure to the cabinet. A detailed explanation of this procedure will be shown in chapter 3.1.8.

Generally speaking, in a linear simulation it is insignificant whether you apply the load in the shown direction or reverse. The result remains the same, but in the opposite direction. Also if the force is set to be half the magnitude, the displacement and rotation (for small angles) will also be half. Only if constraints are depending on load direction like with the “cabinet bolted to a base” case in chapter 2.1.3, the outcome will be different.

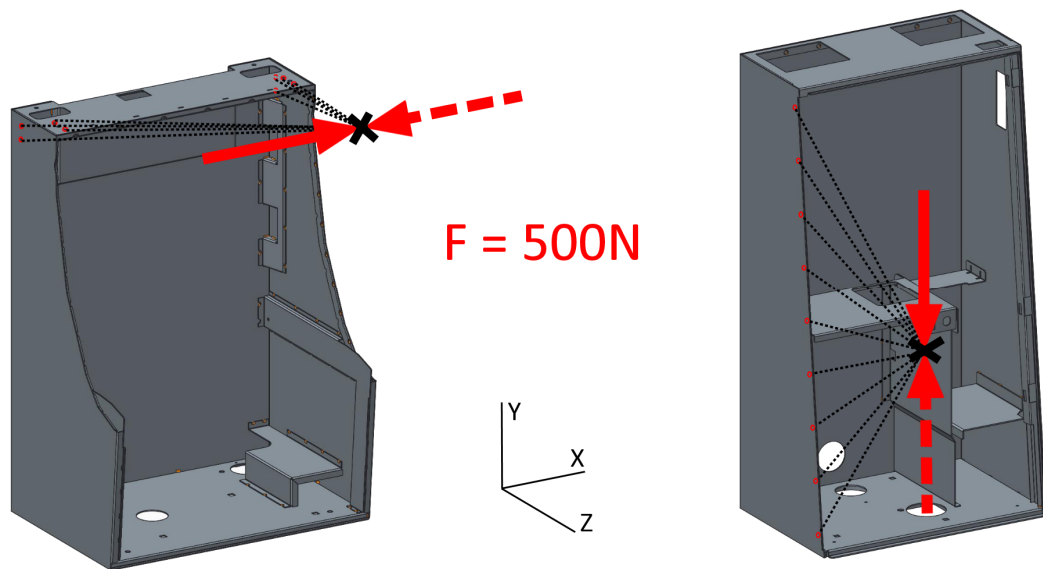


Figure 2-3: Force acting on mounting points

The best basis for a comparison of the amount of deformation is the angle of maximum rotation. Therefore, the angle of rotation about the Y- axis will be analyzed. In this application the angle value of maximum rotation is better suited to compare deformation than a displacement value at any point. Rotation will be comparable between different sizes of cabinets, and it will correlate with the angle the door will be out of place.

2.1.2 Constraints for a Freestanding Cabinet

In addition to applying a load, the cabinet has to be constrained in a proper way. This is a very important task within an FEM simulation, as it will affect the result significantly. The goal is to define constraints that reproduce real world conditions, or better said, reproduce worst-case real world conditions as effectively as possible. The most common mistake in that context is to over-constrain the model so that the simulation is carried out with a far stiffer model than it would be in the real application. The effect of too many constraints will be demonstrated later in this section.

A freestanding cabinet is in most cases not constrained in a well-defined way. Usually it is equipped with four adjustable legs that are sitting on a more or less flat floor, because of the weight of the machine. A certain kind of friction between its legs and the floor keeps it from moving around. While friction is rather low on a hardwood or tile, it will be quite high on a carpet floor. Consequently, the worst-case scenario will be if the legs are free to move around on the floor without any friction, and the legs are fixed only on a minimal level, to hold the cabinet in place and it cannot slip away as a reaction to an external load. This will be achieved by constraining the simulation model in the following way:

One leg is fixed in all directions translational, another adjacent leg will be fixed so it cannot rotate around the first leg but can move away or towards it. The third leg will be free to

move on the ground but is constrained in vertical direction. The fourth leg is not constraint at all (See Figure 2-4).

Further legs are only fixed translational in its center point and are free to rotate about its center of attachment. That makes the simulation more realistic and is yet easily set up. How this is done will be shown in chapter 3.1.7 in detail.

With this set of constraints the cabinet legs and the cabinet itself are as free as possible and can move and rotate freely against each other. As a result the constraints do not add any incorrect rigidity to the model.

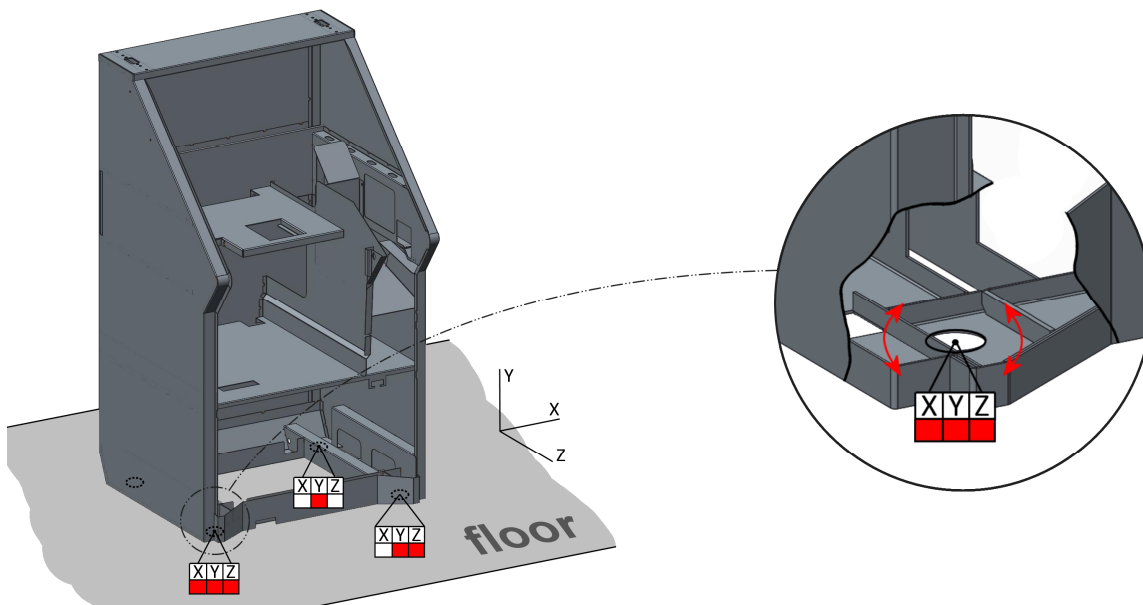


Figure 2-4: Preliminary constraints for freestanding cabinet

Now the question arises whether there is any significance in the fact which leg is fully and which ones are less or not constrained?

Different tests were carried out and the results show dependence on this condition. If there is just torsional moment acting, the results will be the same, but if there is a force applied, it will make a difference (See Figure 2-5).

In this example deformation of the cabinet was simulated with various positions of the defined constraints and the load from chapter 2.1.1. The test shows a significant difference of resulting rotation, if the free leg is either at the front or at the back.

This is a consequence of the fact that in the load cases on the right the force is farther away from the free leg and therefore causes a greater torque at this position, which creates a bigger deformation on the free leg and hence a bigger rotational deformation on the whole cabinet.

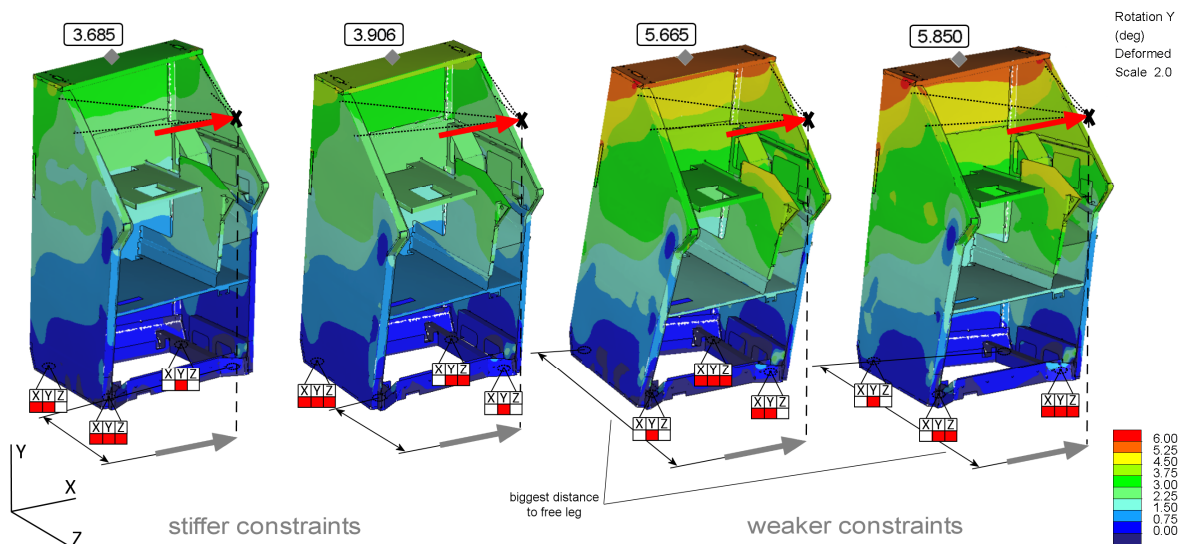


Figure 2-5: Results with different places of constraints

As a matter of this fact, the leg that is farther away from the force should be left free. That makes the cabinet more vulnerable to deformation and represents the worst case. On a more or less symmetrical cabinet it doesn't matter if this will be on the left or right side. Figure 2-5 shows a small deviation between these results, but this deviation is in the range of accuracy of the FEM simulation, and hence doesn't have significance.

Consequently the final system for constraining a freestanding cabinet is determined according to Figure 2-6. Of the constraints shown in dashed lines, the one that is closer to the applied force should be set to get worst case results.

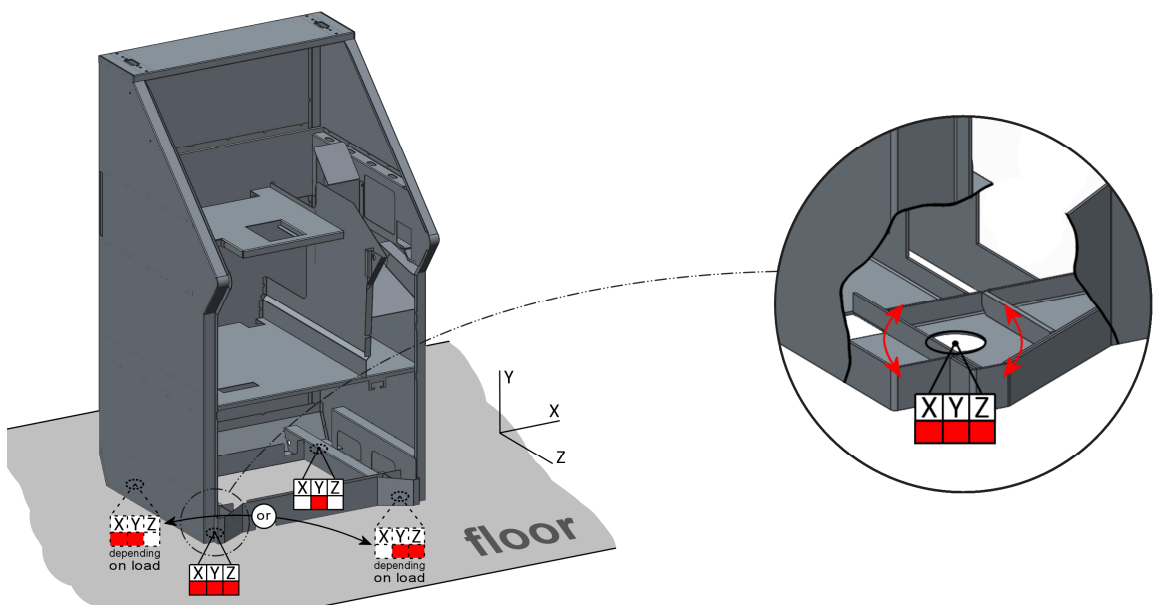


Figure 2-6: Final constraints for freestanding cabinet

To illustrate the effect and importance of constraints generally, two further results with slightly more restrictive constraints are shown as a bad example. Figure 2-7 shows how results are affected. In these cases the twisting of the cabinet is multiple times lower. That emphasizes the importance of proper constraining and the use of the constraint set defined in Figure 2-6.

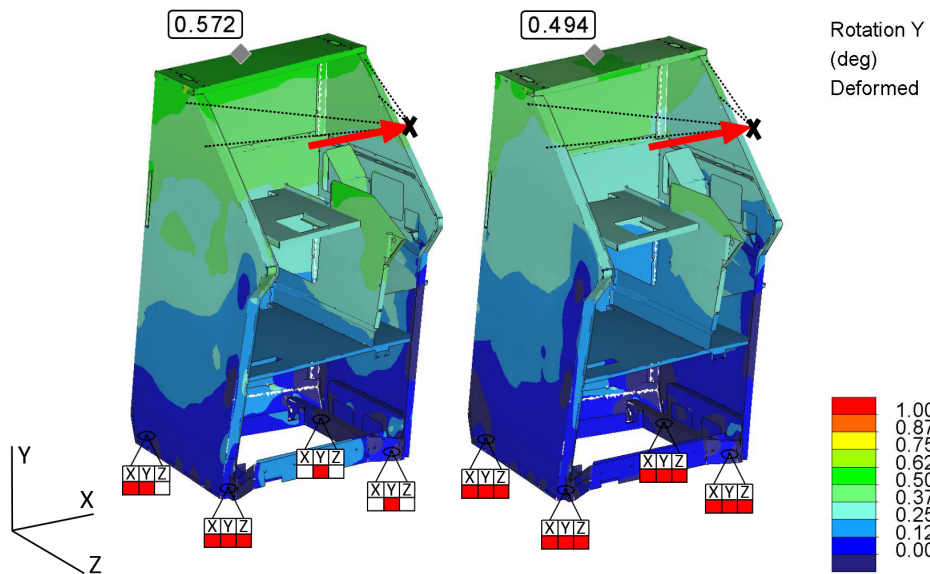


Figure 2-7: Results with more restrictive constraints

2.1.3 Constraints for a Cabinet Mounted to a Base

Most of the gaming machines in a casino environment are rigidly attached to a wooden base - usually with four bolts. That kind of installation potentially provides a lot of rigidity to the cabinet itself. Therefore, the most favorable position for the mounting points is directly in the corners. That means the closer they are to the corners, the better.

Unfortunately, in many cases various requirements obstruct the possibility to adopt this rule properly. Mainly due to the fact that the attachment and the fastening of the bolts has to be done with the machine fully assembled. As a consequence, bolts have to be freely accessible without having to unfasten any components. Therefore, mounting points are sometimes in an unfavorable position. It's important to show their influence in the simulation thus constraints need to be set accordingly.

At first glance setting up constraints for this case of installation is considerably more straightforward than the freestanding installation. Just the displacement of the holes for the mounting bolts needs to be fixed in all directions (See Figure 2-8).

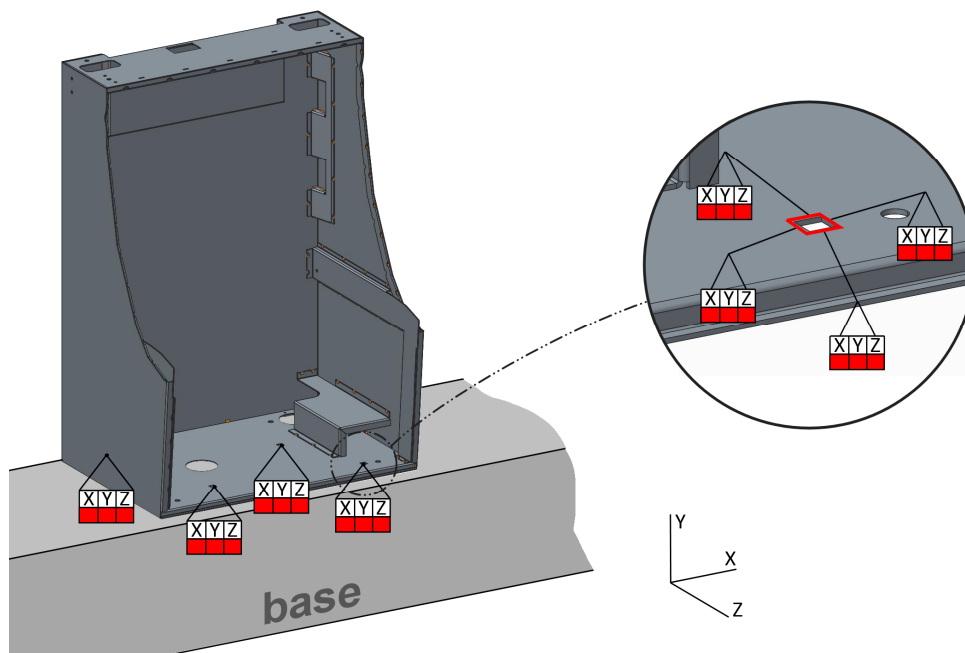


Figure 2-8: Preliminary constraints for a cabinet mounted to a base

But with only this set of constraints the support effect of the base surface is neglected. The fact that the cabinet can't penetrate the base adds some considerable amount of rigidity. Especially if mounting points are located far from the corners. Unfortunately geometry in a FEM analysis cannot be constrained, so it's locked in one direction of an axis but not in the other.

Theoretically, this would only be possible with the help of a contact analysis, but that kind of analysis does only work with solid- and not with shell elements in Creo Simulate. Addi-

tionally it needs to be solved iteratively and therefore it requires a multiple time for calculating the result. So the viable approach to this is setting the constraints manually.

For this reason the cabinet is being analyzed under specific load and the basic constraint set shown in Figure 2-8 is applied. The resulting deformation will be examined to find out which corners are bending into the base. Then these corner points will be fixed in vertical direction. (Fixing points will usually create a point of a stress singularity in a FEM simulation, but as we are only interested in deformation, this is not a problem).

This will provide a result for a specific load case. But if we are interested in results that include forces potentially working in both directions, we need to find out which one will lead to a greater amount of deformation and that requires further investigations.

A load now applied in the opposite direction will not give the same magnitude of deformation like on the freestanding cabinet. This is because the specific constraints in the corner points need to change according to the direction of load. Therefore, both directions of a load need to be analyzed in order to find the most vulnerable for the cabinet. This is done with the help of two different analyses including constraints according to Figure 2-8 and forces in opposite directions. An example of a possible result is shown in Figure 2-9.

After that, the load direction that is most vulnerable needs to be identified. In this figure this is the case in the picture on the left side. The right back corner of the cabinet shows the biggest magnitude of deformation and is free to move away from the base. This is the weak point of the cabinet and the vulnerable load direction. If this point was constrained like in the right picture it would result in a smaller deformation as if the corner with the smaller magnitude of deformation in the left picture was constrained.

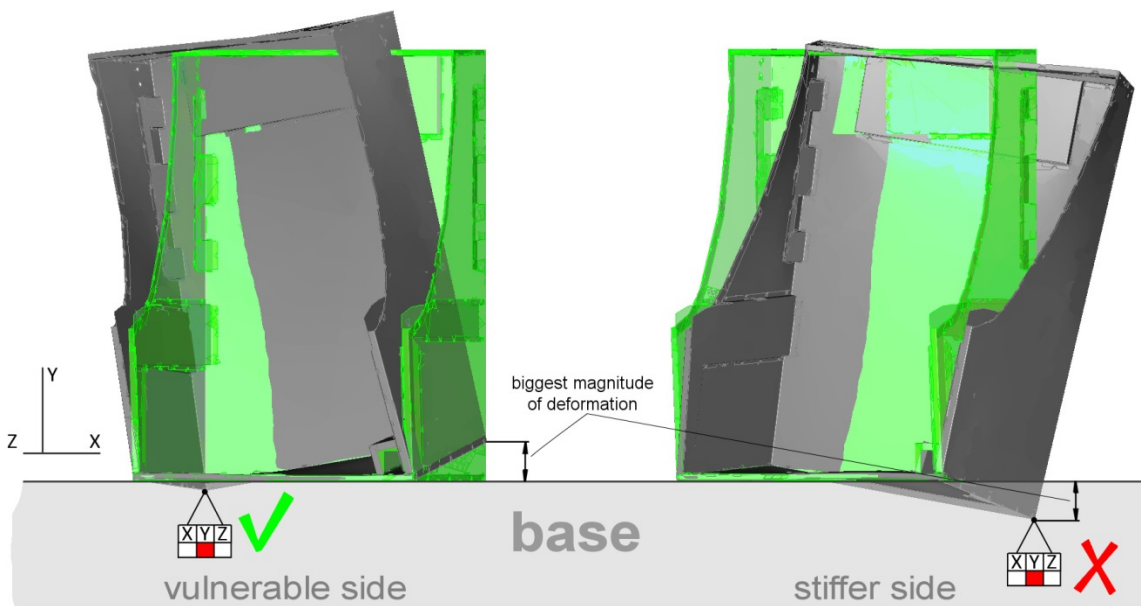


Figure 2-9: Constraints for base support

Therefore, the final analysis should be done with added constraints at the corners of the smallest displacement that all move in the same direction. In cases of doubt, which one is the weaker side; both sides need to be analyzed.

Preferably, during the analysis the right direction of load is kept as well. It is not critical for the resulting amount of deformation, but the deformation will be in the right direction. Figure 2-10 shows the final set of constraints for a base mounted cabinet.

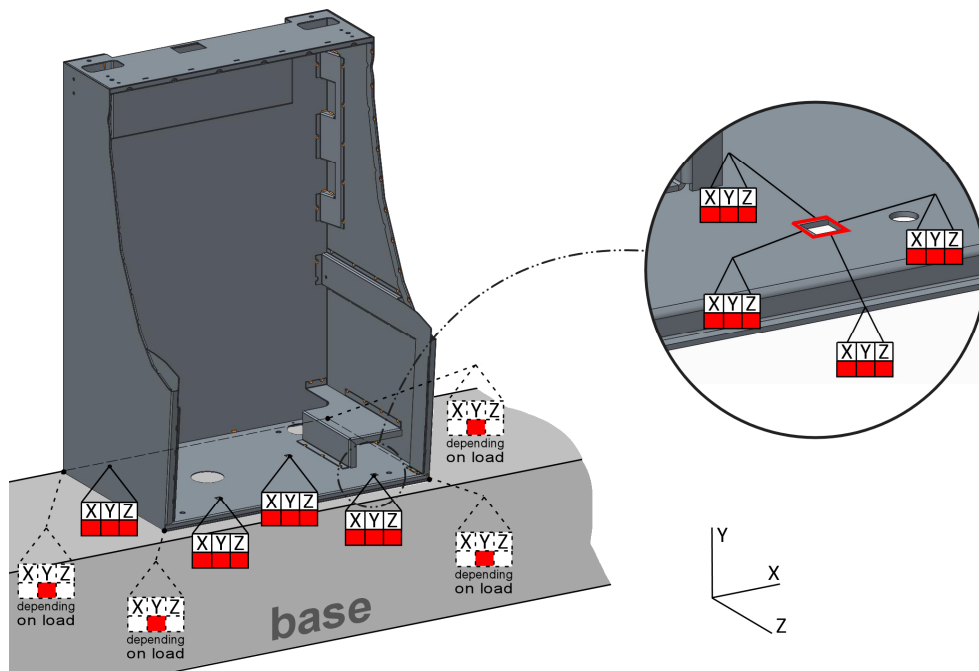


Figure 2-10: Final constraints for a cabinet mounted to a base

2.1.4 Reinforcement with MDF Base Plate

On most of the cabinets mounted to a base, a wooden MDF board is installed between base and cabinet. This board is screwed to the cabinet and increases its rigidity in the bottom area. This is somehow compensating the fact that sometimes mounting points are not at its optimum position. These boards will be attached during the production process, and thus no access during installation in the field is necessary, so it can be screwed close to the corners, which is the best way to enhance strength.

Additionally it closes the gap between the cabinet and the base; it acts as a spacer for the installation bolts and protects the base from scratches.

In order to incorporate the reinforcement effects of the MDF board in the simulation, the board will be idealized as shell element with a separate material definition and “rigid link” connections will be applied between cabinet and board. These connections need to be set where the board is screwed to the cabinet and also on the holes where the cabinet is bolted to the base (See Figure 2-11).

How this is done will be shown in chapter 3.1.

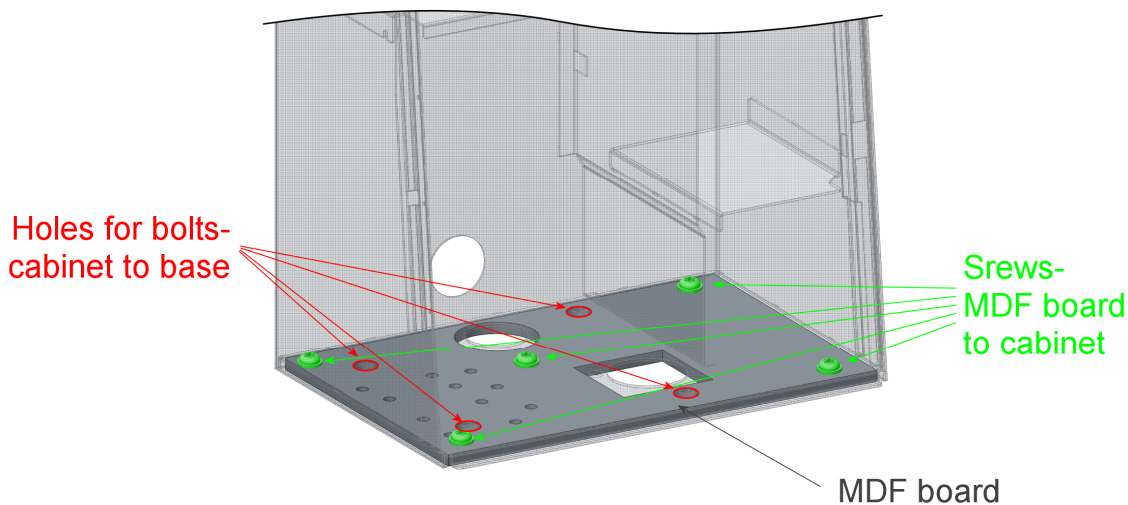


Figure 2-11: Connections for MDF base plate

2.2 Rigidity of a Base Plate of a Free Standing Cabinet

For a freestanding cabinet it is very important to have a rigid base plate. The cabinet will be placed on a floor that is potentially uneven and will bend to try to bring all of its four legs on the ground. This displacement of a leg needs to stay in reasonable magnitude because it will warp the whole cabinet with it and will affect gaps and door mechanisms. Further the legs are usually adjustable in height and therefore should be adjusted to a crooked floor rather than bending the cabinet with it.

To conduct the simulation a force of 500N will be applied on the free leg and the cabinet will be constrained according to Figure 2-12. These constraints are similar to the ones defined in chapter 2.1.2, but it is insignificant where the free leg and the force is located as long as the structural design is effectively symmetric left and right. With a symmetric lay-out of the legs location the reaction forces will be equal to 500N in all point constraints.

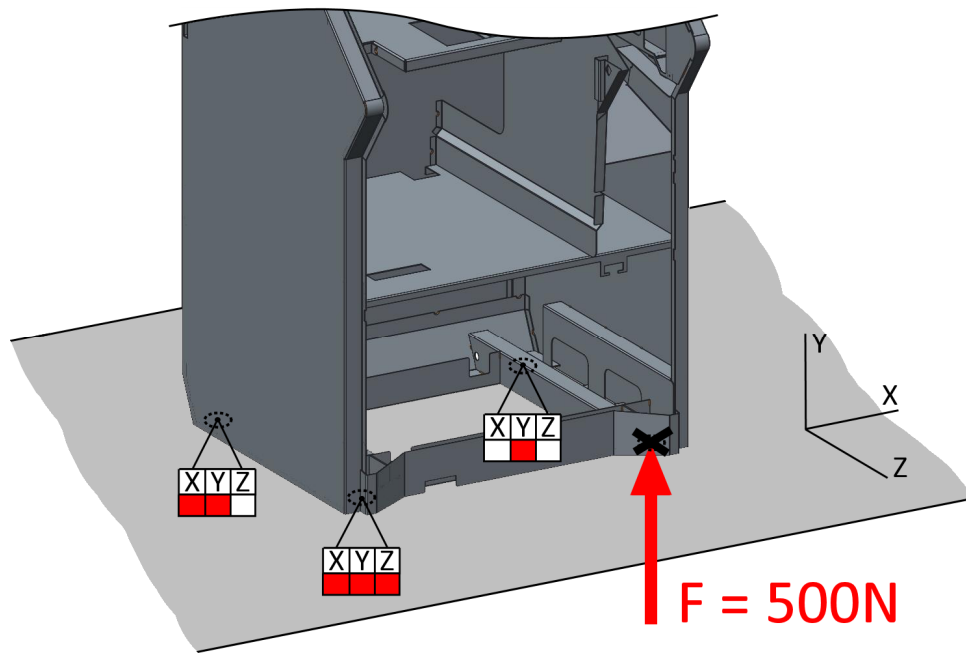


Figure 2-12: Testing force for base plate

The result is defined by the displacement of the outer corner of the cabinet next to the free leg.

This magnitude of displacement is representative for the rigidity of a base plate of a free-standing cabinet. Additionally, it provides a value that will be in a realistic range as the force of weight of the cabinet is usually between 1000-1500N, which will result in a load of around 500N on just one leg if the floor is not flat.

3 Method and Theory

This section explains all the process steps necessary to simulate deformation of a welded sheet metal cabinet with Creo Simulate 2.0. It describes how to achieve representative and revealing results in a quick matter and shows the best procedure for doing pre-processing, execution and post processing steps.

3.1 Preprocessing

3.1.1 Simplified Representation

The first step on the way of preparing a model for a quick and efficient simulation is to remove redundant geometry. All small features, which are not relevant for the result like small holes, clinch hardware or attachment features for cables should to be removed or suppressed (See Figure 3-13).

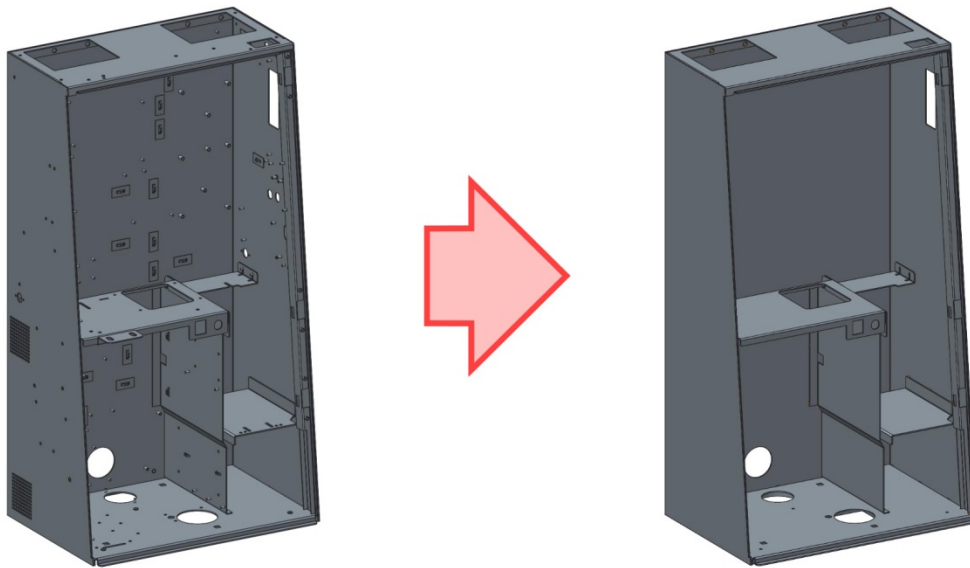


Figure 3-13: Removal of redundant geometry features

Therefore, generally Creo Parametric and Simulate offer a very useful feature called “simplified representation”. In using this feature, geometry of parts and entire parts can be suppressed quickly in a way it is not causing issues with dependencies between them.

Further the simplified state can be saved in the same model and it is possible to switch back and forth between states at any point in the design and simulations can be executed repetitively without having to suppress redundant features again.

Unfortunately in initial tests problems appeared with this feature in combination with the weld connection feature (weld feature is described in chapter 3.1.5). If a simplified representation is used, all weld connections will be ignored and they will not be incorporated in the mesh. Therefore the simplified representation feature can't be used in the course of simulations including weld connections.

This issue was reported to PTC support and is listed under service call C11961792 and their internal SPR (Software Performance Report) 2220845 with priority <HIGH>. The description is the following:

"Creo Simulate): Unable to mesh weld feature when activating a simplify representation in an assembly: after substituting the Model with a his simp Rep a fillet Weld feature got ignored during Meshing and Gives message "Some of Weld connections have been ignored"

As a consequence, until this issue is fixed, redundant geometry needs to be suppressed manually every time before a simulation is conducted. Or a copy of the assembly needs to be created, but this won't stay up to date with changes made to the original assembly.

3.1.2 General Settings

Generally all 3D models in Creo Parametric modeled at GTECH are created in the unit-system $mm\ kg\ s$. Therefore subsequently all values are asked to be entered in these units as default.

E.g. force is entered in $\frac{kg*mm}{s^2}$ by default instead of $\frac{kg*m}{s^2}$, which would be equal to 1N.

But Creo provides very powerful unit handling, so you can enter values in different units throughout the whole system and it will be converted internally. Thus, the general system of units can stay as is and values will be entered in the desired units on demand.

One important step, before defining the boundary conditions for the model, is to set the default interface in the model setup to "Free" instead of "Bonded", else on all coincident surfaces of different parts a bonding interface will be created (See Figure 3-14). That is not desired, as parts should be connected with weld features only.

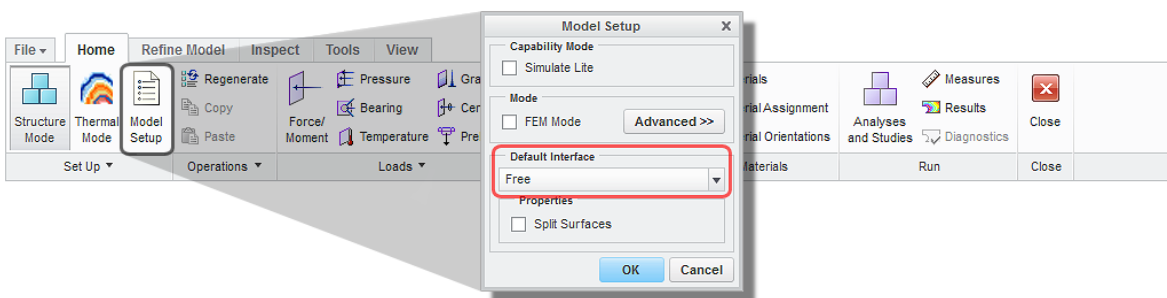


Figure 3-14: Default Interface

3.1.3 Material

Material assignment is generally straightforward. The material STEEL is applied to the entire assembly including all parts (See Figure 3-15). For a possible MDF board that might be attached to a base mounted cabinet (see chapter 2.1.4) a separate material assignment needs to be done. See Figure 3-15 for material properties of MDF and STEEL.

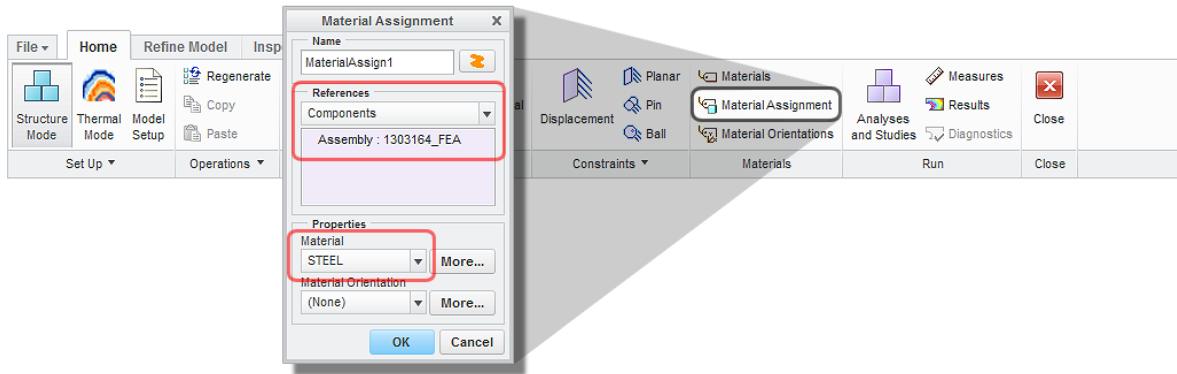


Figure 3-15: Material Assignment

The only properties of importance for materials in a linear static analysis are the Poisson's ratio and the Young's modulus (See Figure 3-15).

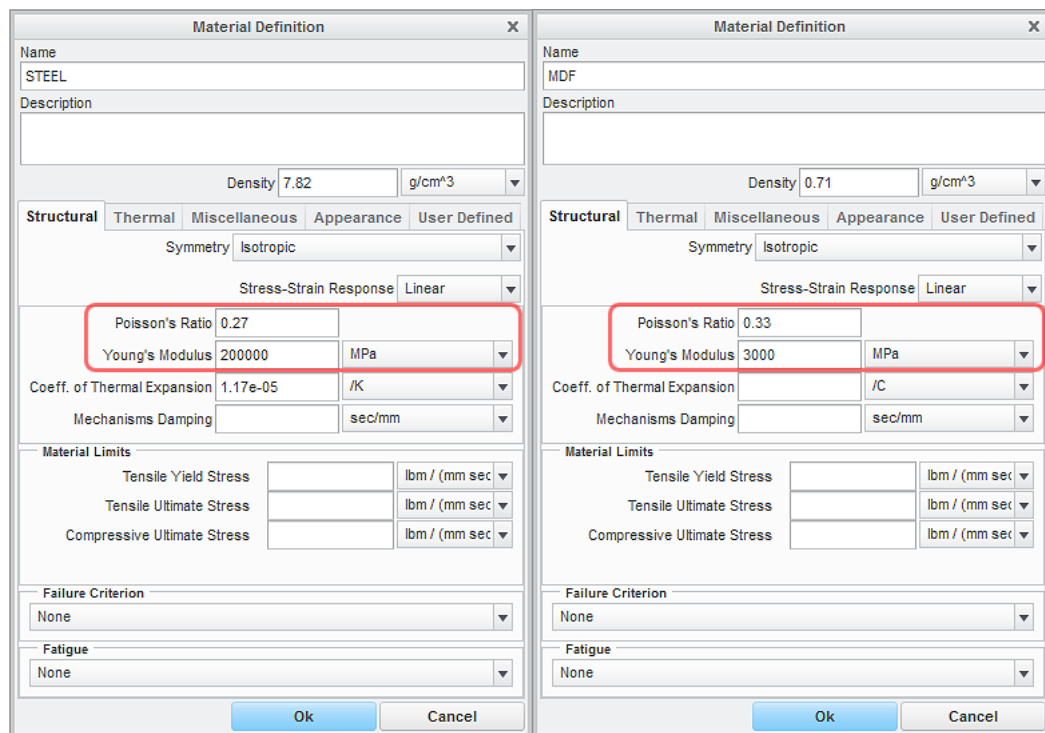


Figure 3-16: Material Definition for STEEL and MDF

3.1.4 Idealization

As indicated in the introduction of this paper it's very important for the efficiency of the simulation to replace geometry of solid sheet metal parts with 2D shell elements. Creo simulate offers a function called “shell pairs”. This idealization method creates a shell feature in the middle of two parallel surfaces and uses the material of the part and the distance of the paired surfaces as thickness property for the newly created flat shell features.

The process of creating shell pairs can be done automatically (See Figure 3-17). The entire sheet metal assembly needs to be picked in the components area. The option “Use Geometry Analysis” can be turned off, if there are sheet metal parts only. Creo Simulate detect paired surfaces that are implicit in some types of features (shells, ribs, sheet metal, thin protrusions) and will convert them to shell elements. In case of having a MDF board modelled as a solid extrusion this option needs to stay checked and a thickness greater than the board needs to be entered. In regards to processing time this option is not critical; geometry analysis is calculated very fast.

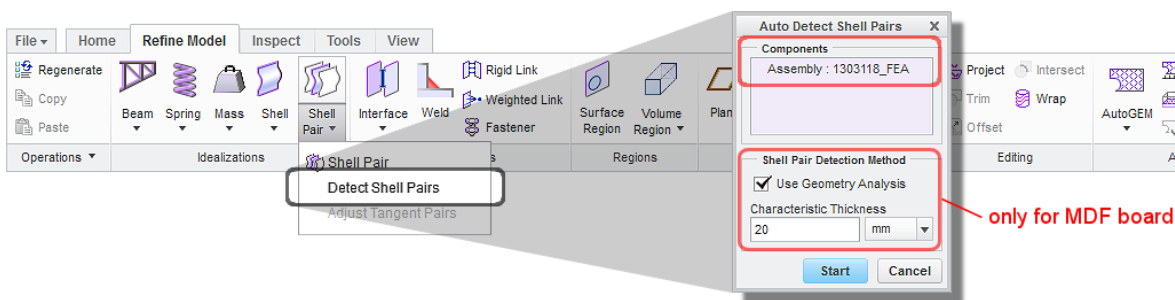


Figure 3-17: Detect Shell Pairs

For an entire cabinet the detect shell pair function potentially creates a few hundred shell features. To check the creation and to review the result the “Review Geometry” function (See Figure 3-18) is very helpful.

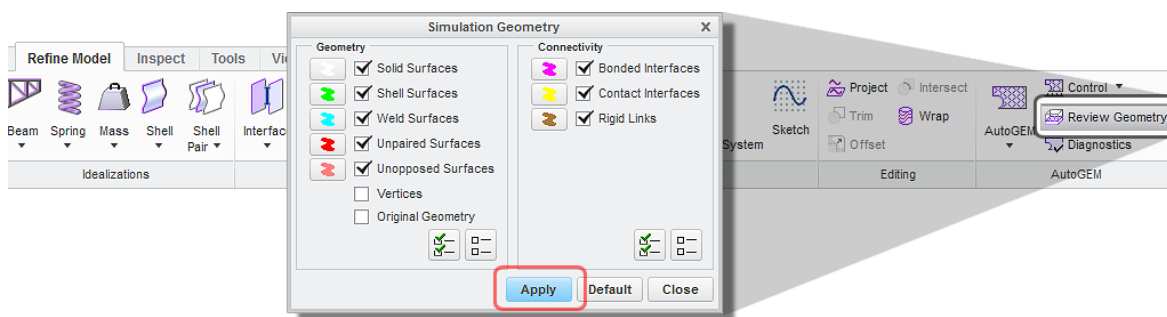


Figure 3-18: Review Geometry

The output of this operation is a representation of the simulation model showing all its simplified and idealized components with connections between them. Shell features are

shown in green. See Figure 3-19 for an example of shell pair representation without any connections.

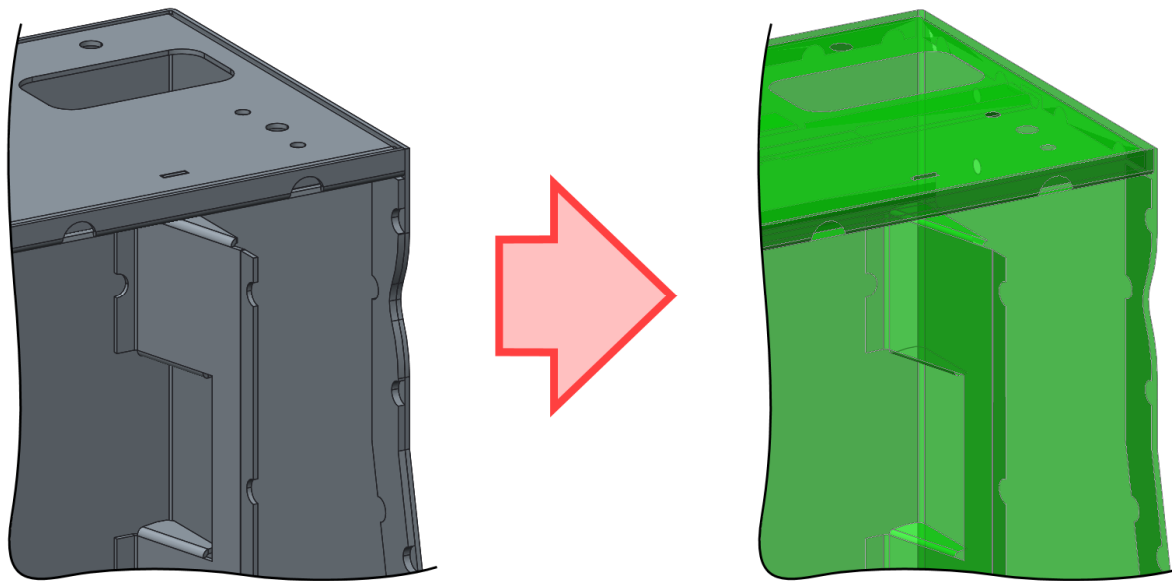


Figure 3-19: Shell pairs derived from sheet metal

This visualization gives the possibility to locate areas of unpaired surfaces or remaining solids in the model. Either of which would cause problems in the process later on.

3.1.5 Connections

Generally in the application of this thesis, all sheet metal parts are connected through weld connections. Therefore, the Creo Simulate module can adopt weld connections defined in the Creo Parametric module and creates appropriate connections in the simulation model. This process can be applied on fillet or groove welds, but not on spot-welds.

If spot-welds are used, they have to be defined once again in the simulation module, but can utilize existing datum points to support the placement. This dedicated spot weld feature is also very helpful if a screwed connection need to be incorporated in the model. It creates a circular connection between two parts and can quickly represent a simplified screwed connection.

For the adoption process of fillet or groove welds from CAD models, weld features in Creo Parametric need to be modeled with the option “surface” (See Figure 3-20).

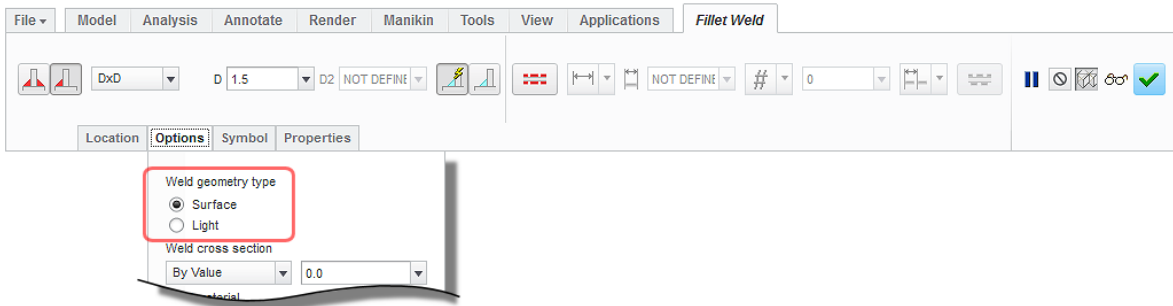


Figure 3-20: Weld geometry type

The weld feature in Creo Simulate will create shell elements of weld line surfaces from the CAD model and project these surfaces onto the middle surfaces of shell pairs of sheet metals. How this is done is shown in Figure 3-21.

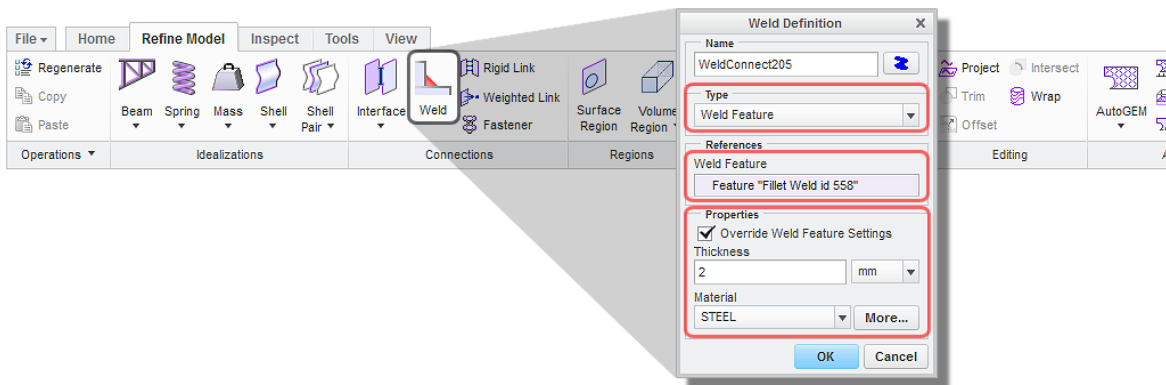


Figure 3-21: Weld feature adoption from CAD model

This adoption process needs to be done feature by feature. That means, the more weld lines are combined in one feature in the CAD model, the less features need to be created in the simulation model. Anyhow with the *CTRL+C* and *CTRL+V* function of Creo a great number of weld features can be adopted relatively quickly.

Further a material and thickness needs to be defined for the weld shell element in some way. Generally this information can be taken from the CAD model if a weld material is assigned to the weld line feature. But defining the material there is generally not part of the procedure. Thus, it is a faster approach to override thickness and material properties of the weld element while adopting it from the CAD model as shown in Figure 3-21.

In case of a possible MDF board attached to the bottom of a base mounted cabinet, this board needs to be connected to the sheet metal cabinet somehow as well. The easiest this is done in applying a rigid link connection between the edges of the screw holes of the cabinet and the MDF board (See Figure 3-22).

Why this has to be applied to the edges and not the surfaces is described in chapter 3.1.7 (Constraints) as are further details on rigid links explained too.

A dedicated rigid link feature has to be created for every individual hole of mounting bolts and every screw according to Figure 2-11 in chapter 2.1.3.

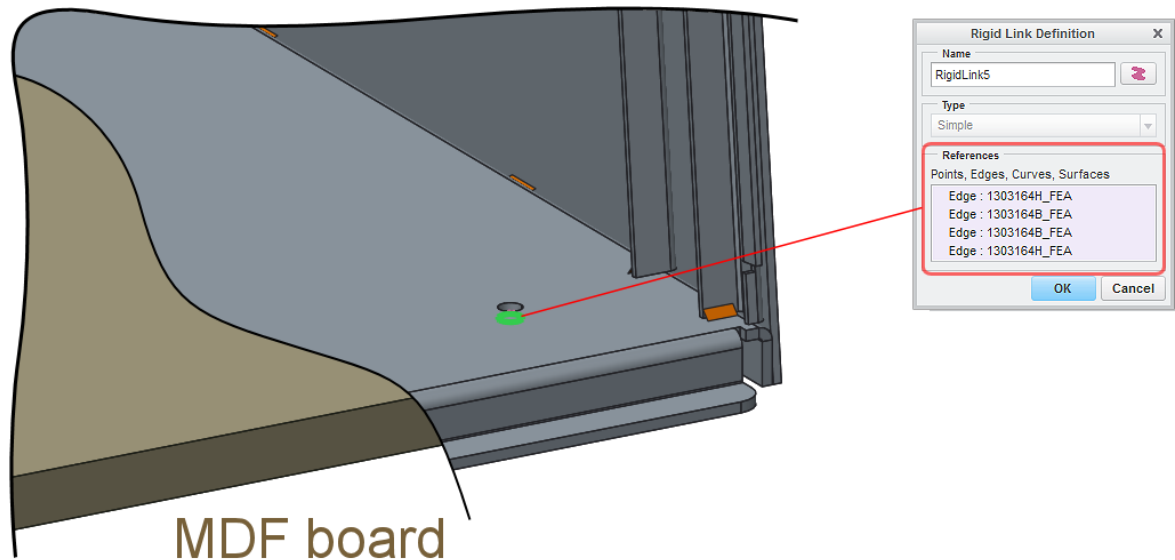


Figure 3-22: Connections for MDF board

As last step, connections should be reviewed if they are successfully created. This can be done like the shell pairs with the “Review Geometry” function in the AutoGEM area (See Figure 3-23). This picture shows a perfect result that is free of errors.

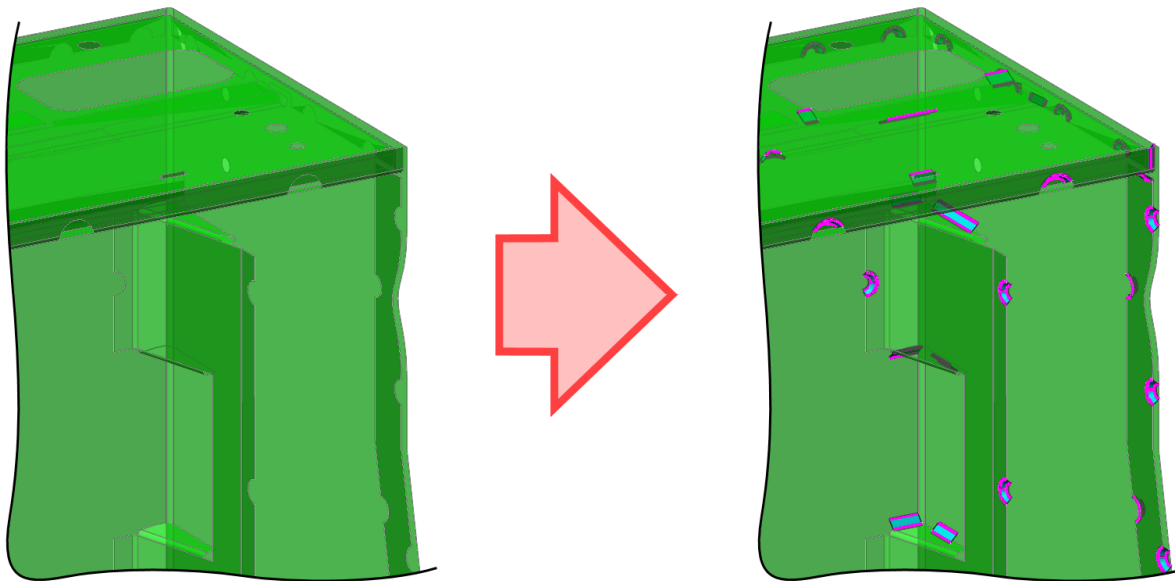


Figure 3-23: Weld connections derived from CAD model

However, sometimes surfaces, when projected onto paired surfaces, produce corrupted geometry. This is often happening when weld lines are placed in small holes or along small rounds. In the event of this, weld line thickness or other geometry properties need to be changed slightly in order to solve the issue. Figure 3-24 shows how a corrupt and a corrected weld-connection looks like.

Before proceeding, all weld connections should be verified in the “Review Geometry” view in order to find problems like that, because if they appear first during mesh creation, issues are a lot harder to find.

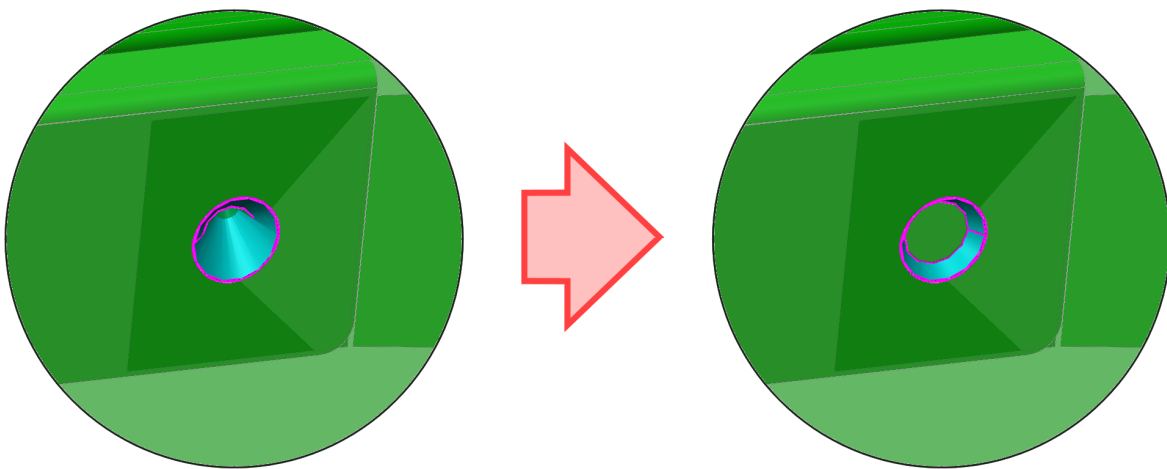


Figure 3-24: Common problem of weld lines in small holes

3.1.6 Mesh

Creo Simulate creates finite elements in a mesh different to most other FEA programs. A lot of other simulation programs are following the classic approach with using h-elements, which use low order interpolating polynomials in each element. This has significant effects. In h- elements the interpolation of the displacement between the nodes of an element is typically linear (first order). Therefore the strain and stress in the element is constant everywhere. This leads to inaccurate values especially for strain and stress, but also inaccurate values for deformation, if big and non-uniformly shaped elements are used.

Creo Simulate is following a different approach with using p- elements (“p” stands for polynomial). A mesh with higher order elements requires fewer elements in order to give an accurate result. Therefore elements can be long and slender or transitions in element size through the mesh can be abrupt. This difference in size will be compensated with increasing the polynomial order on certain elements.

Meshes with mentioned element types giving comparable results are shown in Figure 3-25.

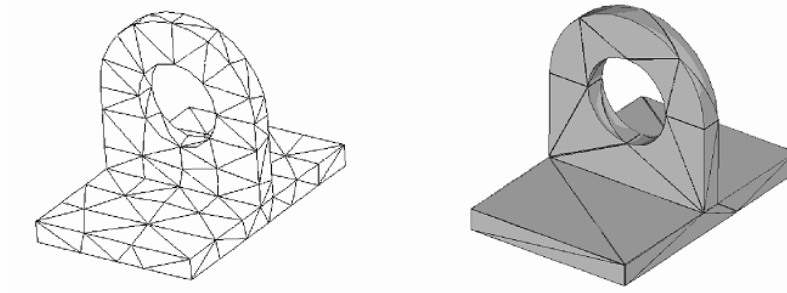


Figure 3-25: Mesh with h- elements (left) and p- elements (right) [ToRo2012]

Creation of the mesh will happen automatically during execution of the analysis. In the course of this, standard settings for element creation work well for most cases. However in order to avoid issues it is good to change a few of these settings.

As first action, AutoGEM can be told to only mesh middle surfaces of shell pairs. All solids remaining in the model will be ignored. How this is set is shown in Figure 3-26.

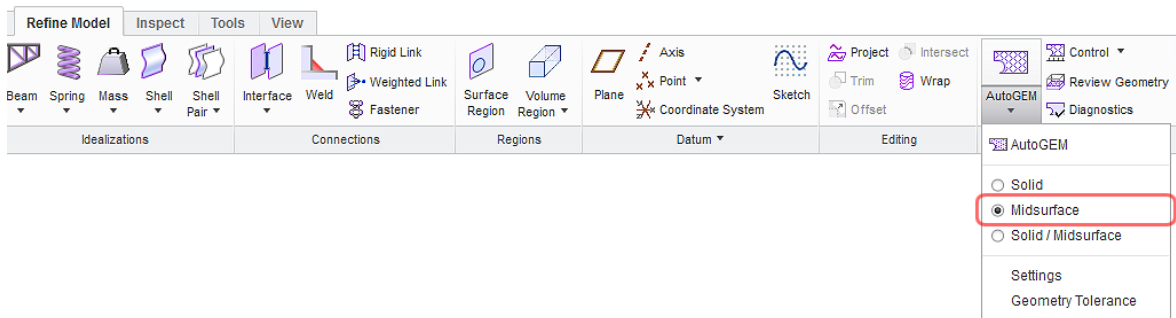


Figure 3-26: Midsurfaces only

Further in the AutoGEM settings (See Figure 3-27) the box “ignore unpaired surfaces” should be checked. This effects, if shell pair detection is not able to create middle surfaces for some odd shaped features, they will be ignored and don’t cause the simulation to fail. Sometimes this can happen on very small sheet metal features that are not relevant for the result. Anyhow, ignored objects should always be monitored whether they are important for the result or not.

In certain cases it will happen that default element size is too big to achieve sufficient convergence. This is due to the fact that default settings are optimized for solid elements creation. Therefore, for shell elements, it is best to tighten rules for element creation a bit. This will make it easier to obtain convergence in reducing the required polynomial order of elements. See Figure 3-27 for recommended settings from PTC [ScHe2014].

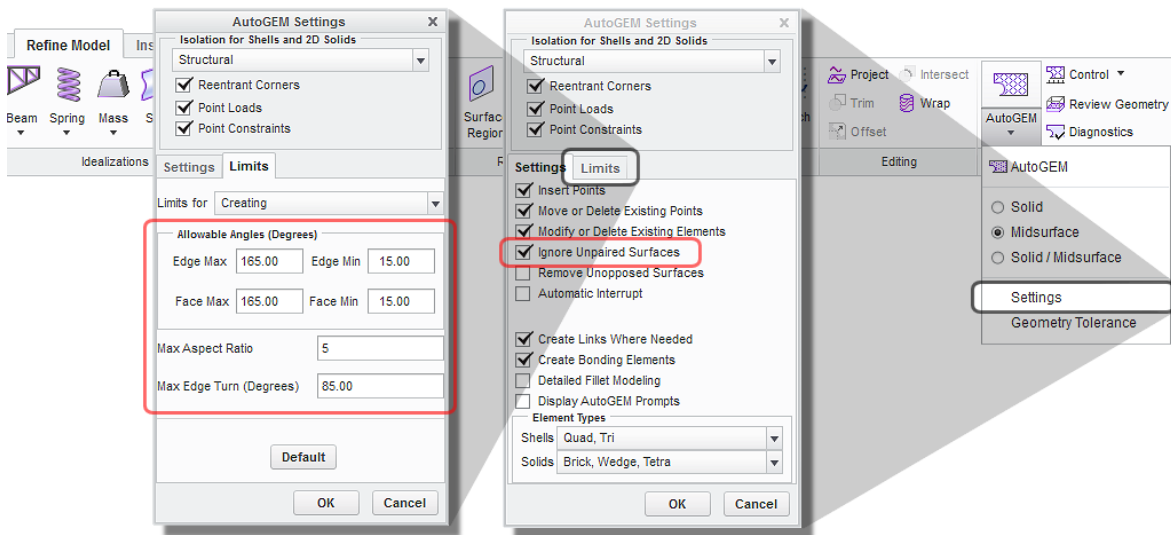


Figure 3-27: AutoGEM settings

After general settings are applied and shell pairs and weld connections are defined, it is best practice to manually create the mesh to check if any problems appear. Sometimes AutoGEM has issues in creating a mesh for certain geometry features. This will be shown during its creation and troubleshooting is easier in this module than during the analysis run. Therefore go to AutoGEM -> create. This can potentially point out first problems, but only the “validate mesh” function will show all critical problems (See Figure 3-28). In some cases it might be necessary to change the geometry slightly in order to get a flawlessly created mesh.

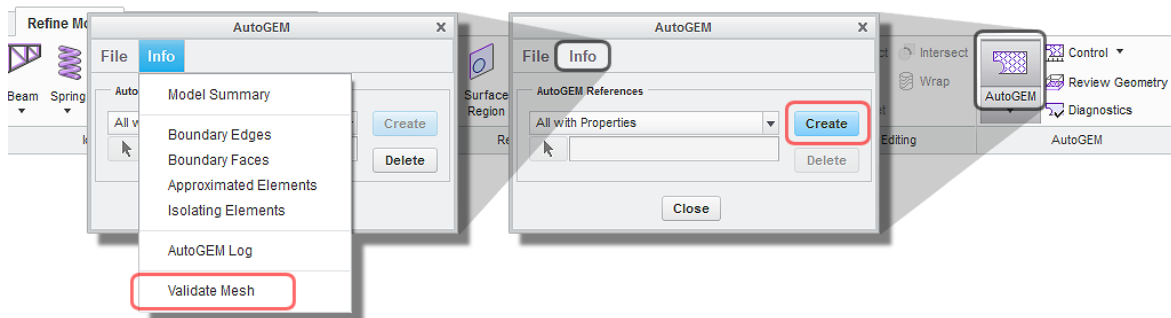


Figure 3-28: Create and validate mesh

Creation of the mesh will be performed automatically with triangles and quads (See Figure 3-29). Local mesh control or mesh refinement is not necessary in the course of deformation analysis.



Figure 3-29: Mesh

3.1.7 Constraints

In 3D solid models, constraints are normally applied to surfaces. A constraint or load applied to an edge or point will cause infinite stress or a stress singularity at the nodes of related elements, because a force or moment is applied to a surface area with zero size.

Working with shell elements, constraints or loads placed on small surfaces on the sides of elements will disappear as surface pairs will be substituted by midsurfaces. Instead, constraints or loads need to be placed on edges or points or on one of the collapsing surfaces itself. In doing so, it doesn't matter whether it is applied to the top or the bottom surface – all boundary conditions will collapse to the same midsurface.

Generally, with shell elements, constraints on points will lead to stress singularities. If they are applied on edges they don't. Therefore, if stress is of interest constraints and loads are preferably added to edges of shell elements.

Further it should be noted that, unlike solid elements, constraints on edges and points of shell elements have rotational degrees of freedom. That will be discovered in settings for constraints later in this section.

Freestanding Cabinet:

Usually constraints will be applied directly to elements of the model. However, the defined constraint set for a freestanding cabinet (Shown in Figure 2-6) shows a constraint added to the middle of the mounting hole of a cabinet's leg. This has the benefit that the point is only fixed in position but not rotation.

This is achieved in creating a reference point in the center of the respective hole first and then a rigid link that connects this point with the edges of the mounting hole for the leg (See Figure 3-30). This results in, as the name rigid link implies, a rigid connection between references that doesn't allow any displacement between them. As a consequence the references itself, in this case the circle halves, are fixed in shape as well.

As a matter of the fact that a rigid link between all selected references in one feature will be created, but the individual legs should stay independent from each other, several discrete features need to be created – a separate one for each leg.

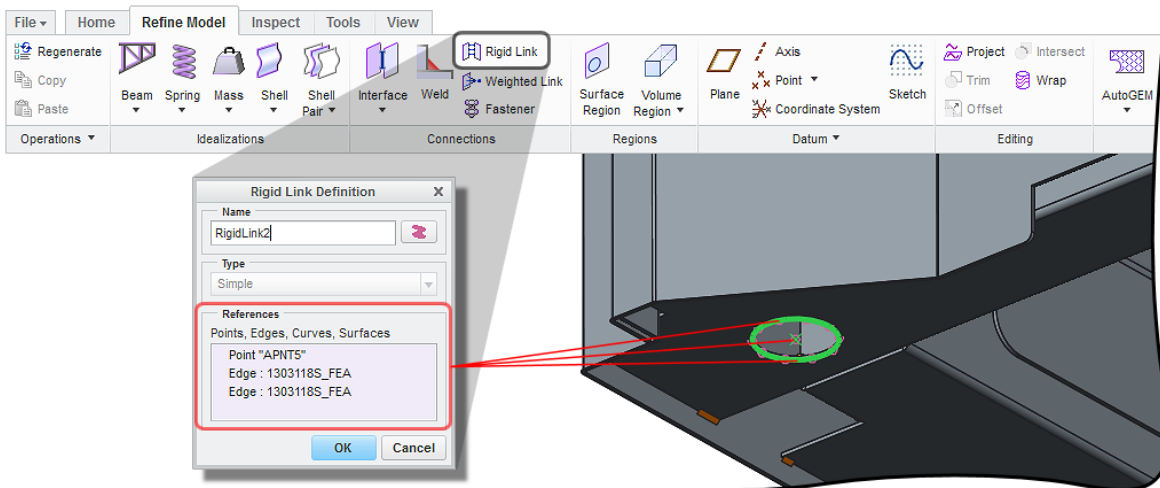


Figure 3-30: Rigid Link for cabinet leg

The points itself are then constrained with displacement constraints (See Figure 3-31). They are fixed according to the system defined in chapter 2.1.2 and Figure 2-6. Depending on that, translation is fixed or free in respective direction.

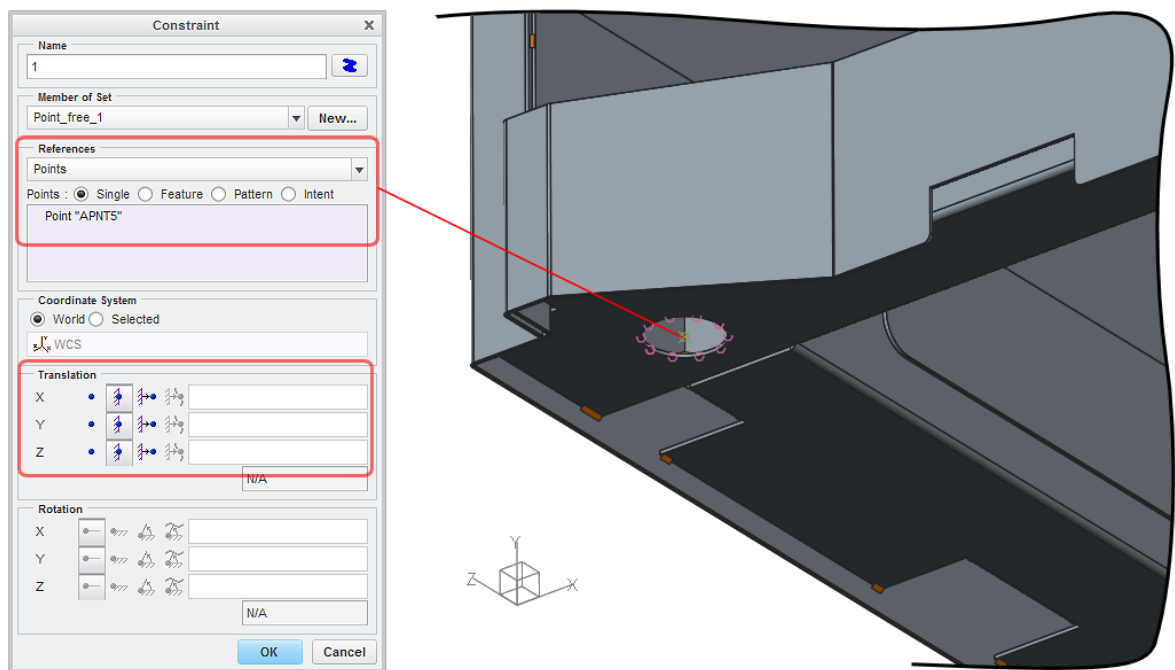


Figure 3-31: Displacement constraint for freestanding cabinet

Cabinet Bolted to a Base:

Figure 3-32 shows the constraint set for fully constrained mounting holes. All edges of the holes will be fixed translational and rotational in all directions and about all axes. The constrained rotation represents the situation that the head of mounting bolts keep the surfaces around the edges plane.

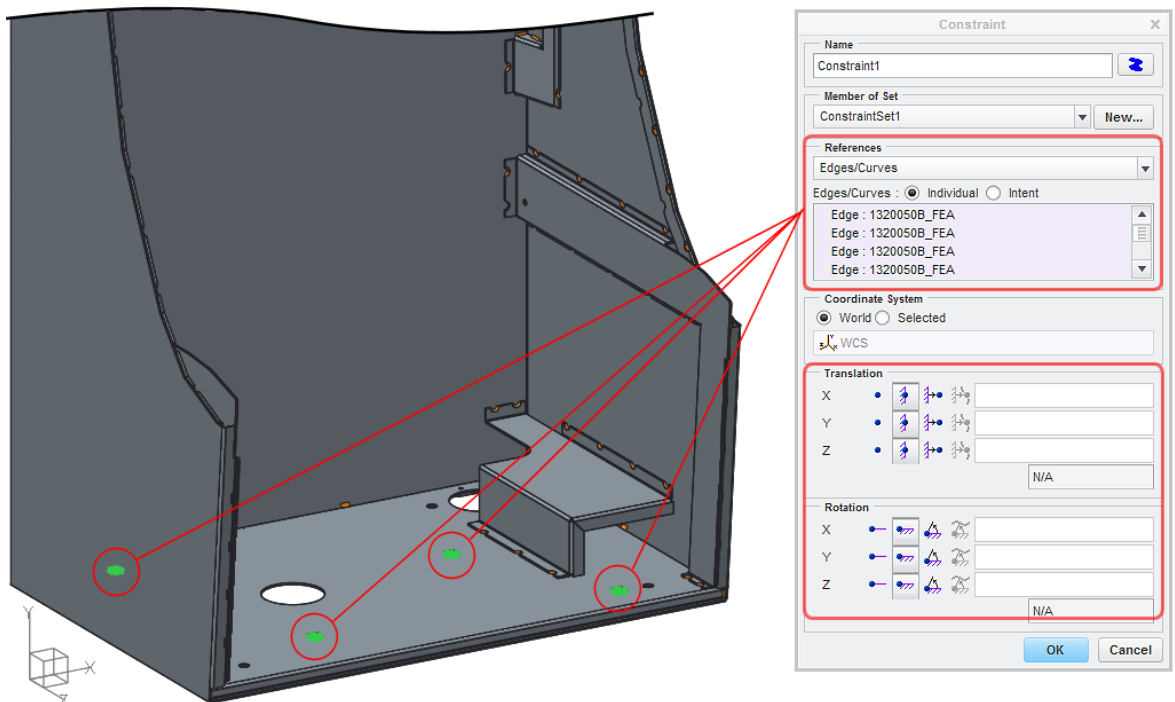


Figure 3-32: Fixed mounting bolt holes

Figure 3-33 shows how to set up the point constraints, which are defined in chapter 2.1.3 and represent the support the base provides when the cabinet is attached to it. According to the definition made in this chapter, one or more points need to be constrained in Y (vertical) direction. Rotational freedom will be left free.

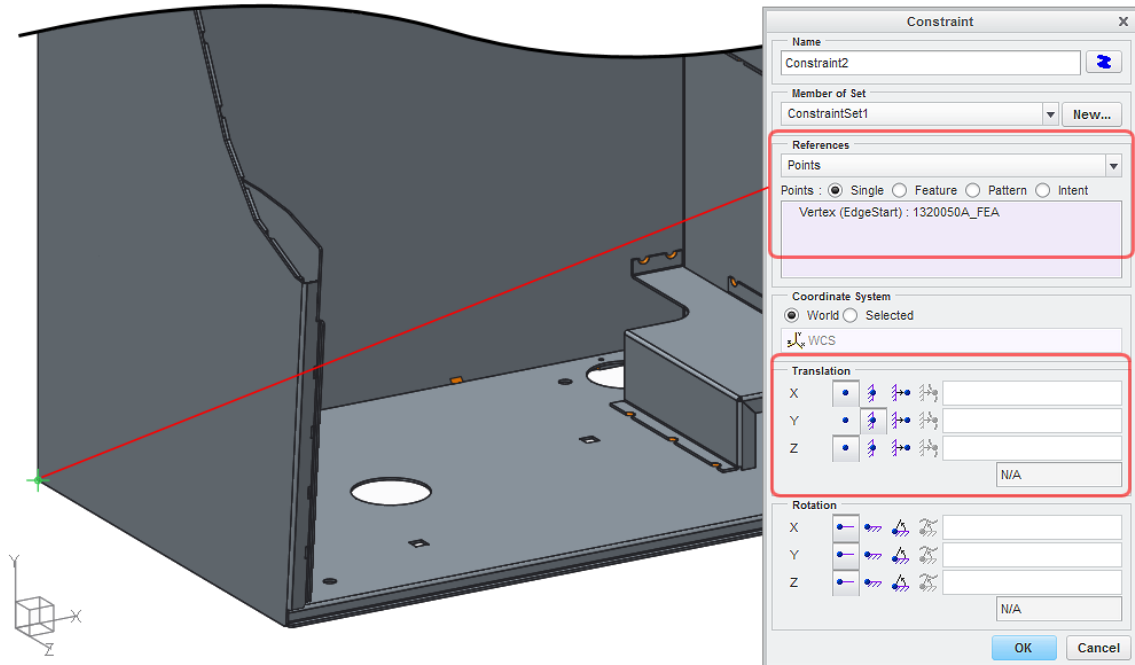


Figure 3-33: Point constraints for base support

3.1.8 Loads

In order to apply a load according to defined scenario in chapter 2.1.1, the “Total Load at Point” option needs to be used. This function is available in the Force/Moment Load dialog box in Creo Simulate with a click on “Advanced” (See Figure 3-34). With this option it is possible to distribute a load on an edge or surface that is equivalent to a load applied to a single point.

First a datum point on the designated spot needs to be created. According to chapter 2.1.1 this point lies on the most forward point of the door. The load is now applied to this point and, as shown in Figure 3-34, distributed to the selected geometry in the reference tab. Like with setting up the constraints, only edges should be picked here.

The load is set with 500N in the negative X direction, as an example in this case, and the unit is changed to Newton. Like explained before, direction of load only affects the direction of deformation but gives same magnitude of deformation.

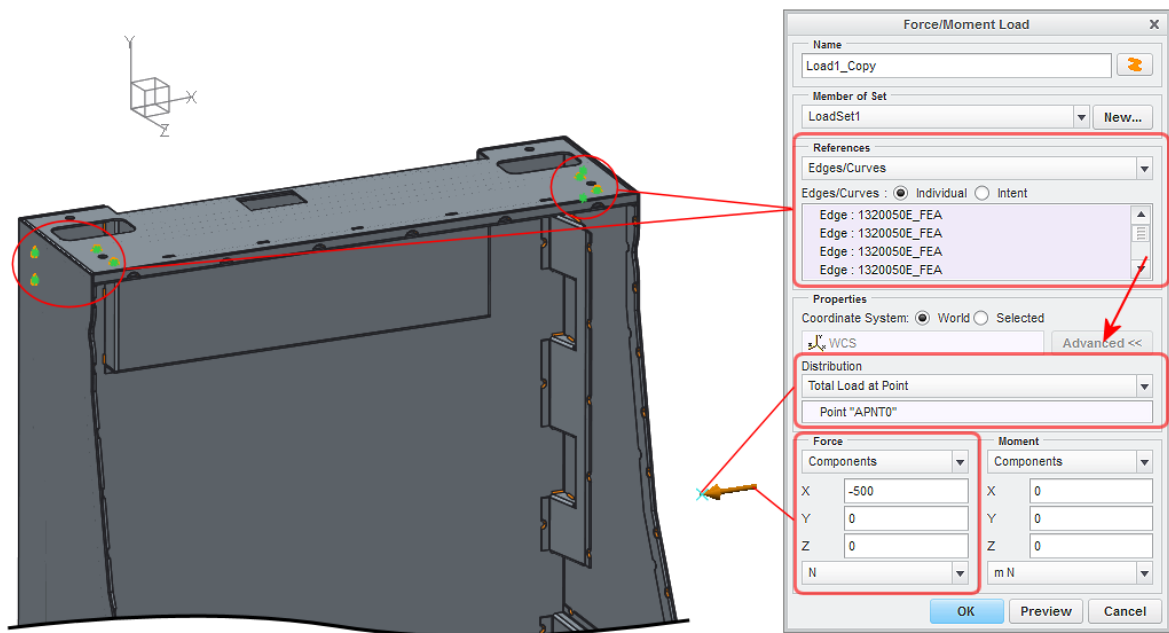


Figure 3-34: Load on virtual point

3.2 Executing the Simulation

3.2.1 Static Analysis

After all preprocessing steps defining the FEA model are completed; the actual analysis can be created. Figure 3-35 shows the creation of a new “Static Analysis” and the basic settings. Most important is that the right constraint and load sets are picked.

It's possible to pick one constraint set but multiple sets of loads that will be calculated in the same run. When viewing the results, these loads can be evaluated separately or as a random combination of different loads. This is an interesting feature if results from more than one load are of interest.

Further, as shown in the “Output” tab in Figure 3-35, special attention should be paid to which properties will be calculated during the analysis solving process. Important in this context is that calculating stress takes twice to five times longer than calculating just displacement and rotations, depending on the method for obtaining convergence.

Therefore, as stress is only of secondary interest in this investigation, it should be considered to be left out for most of the analysis runs and only added when really desired.

A very important step during the setup of an analysis is defining the convergence procedure. This is also done in the present dialog, but will be addressed separately in the next chapter 3.2.2.

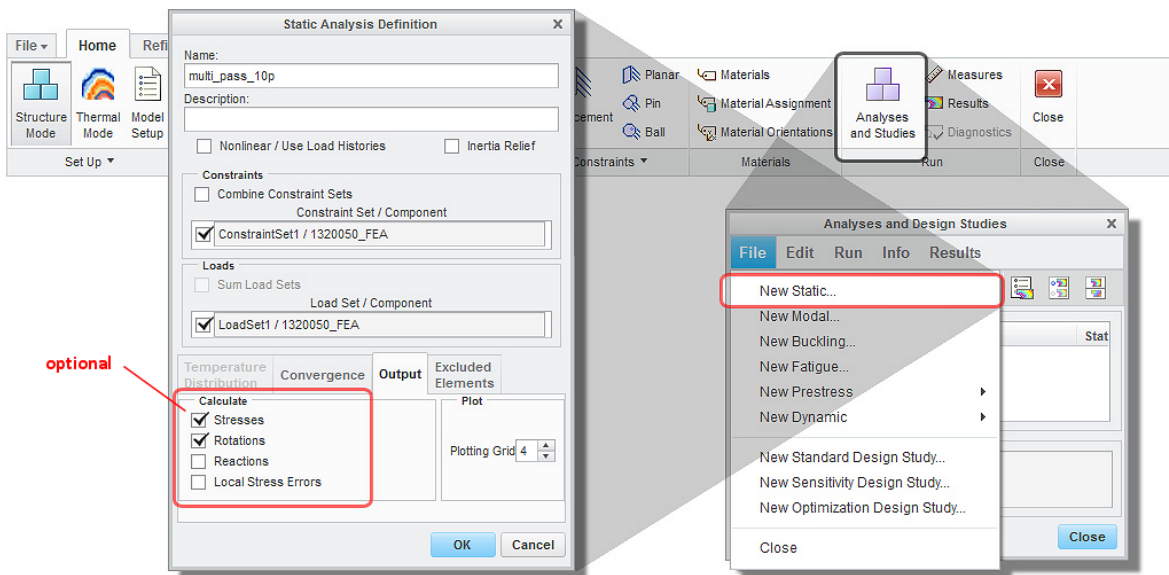


Figure 3-35: Static analysis

Concerning general settings for the analysis run, there are a few that are of special interest. As shown in Figure 3-36, memory allocation should be set to half the size of the physical RAM. The standard setting is just 512 MB, which is very little considering configurations of current workstations and can cause problems even on models with a small number of elements. The shown case is for a PC with 16GB of RAM.

Another setting in this dialog can be interesting: “Directory for Temporary Files”. In all analysis calculated during the investigations for this thesis, neither the CPU nor the amount of RAM was the determining factor or bottle neck for the calculation time. It was the speed of the hard drive for the temporary files. That means if there is a fast solid state drive in the workstation this should be used for the temporary folder.

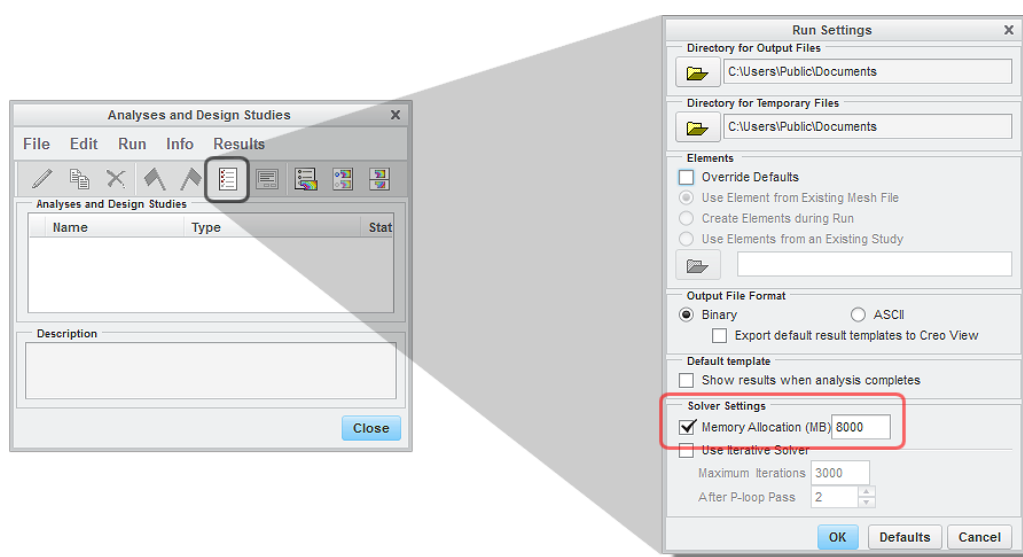


Figure 3-36: Settings for static analysis

Finally the analysis is run with pressing the green flag button (See Figure 3-37). In the course of this a question pops up, if interactive diagnostics should be run as well. If a model with specific settings is analyzed the first time, it is very important to enable this function. It will show potential problems that will let the analysis fail in red and potential issues that need to be reviewed in regards to its potential impact on the desired result in orange color.

As an example, rather obvious issues or problems that can appear or will be noted are, parts are not connected to any other parts, the model is not sufficiently constraint, existence of possible stress singularities or certain elements are ignored, because of rules defined in the settings.

More troublesome issues are for example if the geometry causes issues with meshing the model. In this case only changes of the AutoGEM settings or slightly changed geometry of the respective part can help solving the problem.

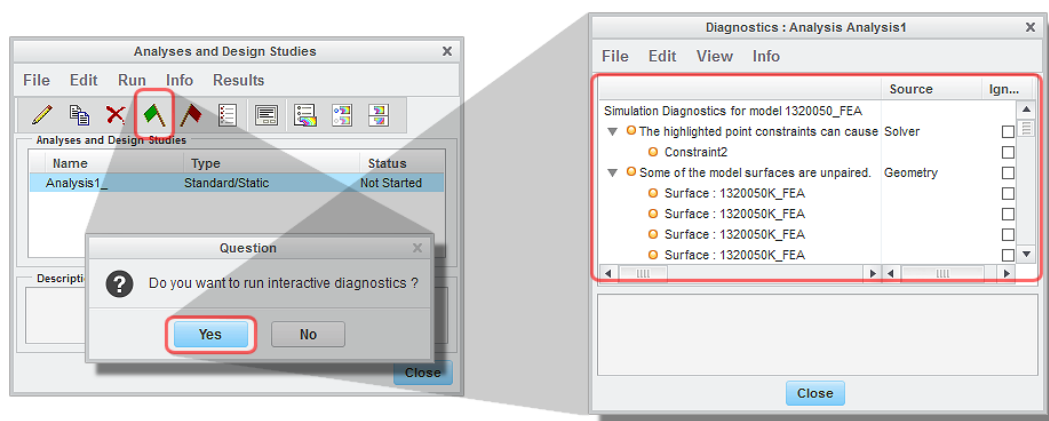


Figure 3-37: Diagnostics for analysis

3.2.2 Convergence

As already discussed in chapter 3.1.6 for the mesh, Creo is using p- type elements instead of h- type elements, like many other FEA programs, when it comes to convergence analysis. The “h” in the name of h- elements is borrowed from the field of numerical analysis, where it denotes the fact that accuracy is related to the step size used in the solution, which is usually expressed by “h”.

Creo Simulate is following a different approach with using p- elements (“p” stands for polynomial). The main difference to h- elements is that in order to achieve better and finally sufficient accuracy the mesh doesn’t need to be recreated with finer and finer elements. Instead, convergence is obtained by increasing the order of the polynomials that are interpolating the displacement between the nodes on each existing element.

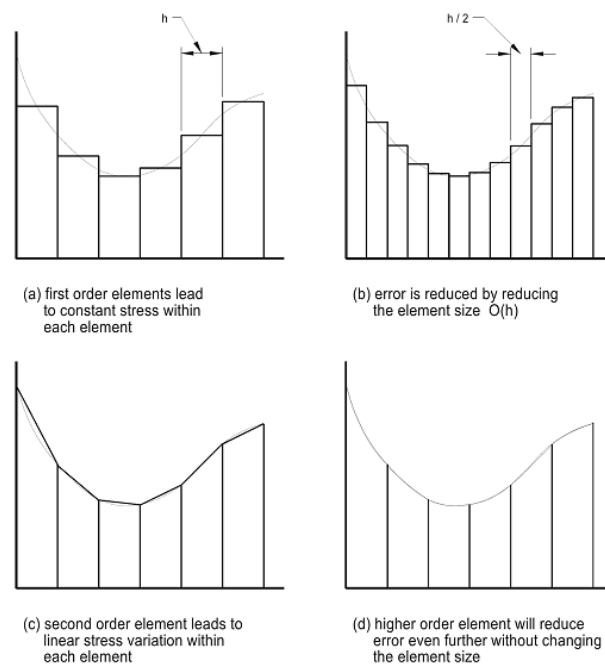


Figure 3-38: Convergence with h- elements (top) and p- elements (bottom) [ToRo2012]

As illustrated in Figure 3-38, the main benefit of this method is, the same mesh can be used throughout the convergence analysis, rather than recreating meshes with smaller elements required by h-elements. This results in less CPU time of the calculation and potentially less problems with mesh generation.

During the convergence process, Creo Simulate only bumps up elements in regions of high gradients to higher order polynomials. By examining the effects of going to higher order polynomial, Creo can monitor the expected error in the solution, and automatically increase the polynomial order only on elements where it is required. This is implemented with three different options:

- **Quick Check (QC)** – This is not really a convergence method as the model is only solved for a fixed, low polynomial order of 3 for all elements. Thus the result should never be trusted. Main reason to use this method though is to pick up any errors that may have been made and to verify constraints and load in reviewing if the model is behaving the expected way.
- **Single Pass Adaptive (SPA)** – It's a convergence run of just two cycles. The first run is done with low polynomial order elements. Then the accuracy of the solution is assessed by Creo, the p-level of certain critical elements is modified and a final pass with raised order of certain elements that should provide reasonable results is done.
- **Multi Pass Adaptive (MPA)** – This method gives the best control of the convergence process. Multiple passes are made through the solver, while increasing the polynomial order on certain elements by one or two with each pass. This iterative approach continues till the solution converges to a defined accuracy or the maxi-

imum polynomial order is reached. Default measures are displacement, strain energy and global RMS (root-mean-square) stress, but any other measure defined or predefined can act as criteria.

With the MPA method the desired result at the desired spot can be monitored in its accuracy. A case in this thesis is the resulting rotational deformation at the top of the cabinet. To monitor this result during the convergence process, on this spot a “Measure” feature needs to be created. See Figure 3-39 how this is set up. The unit “rad” can’t be changed to degree in this dialog, but this can be done in post processing.

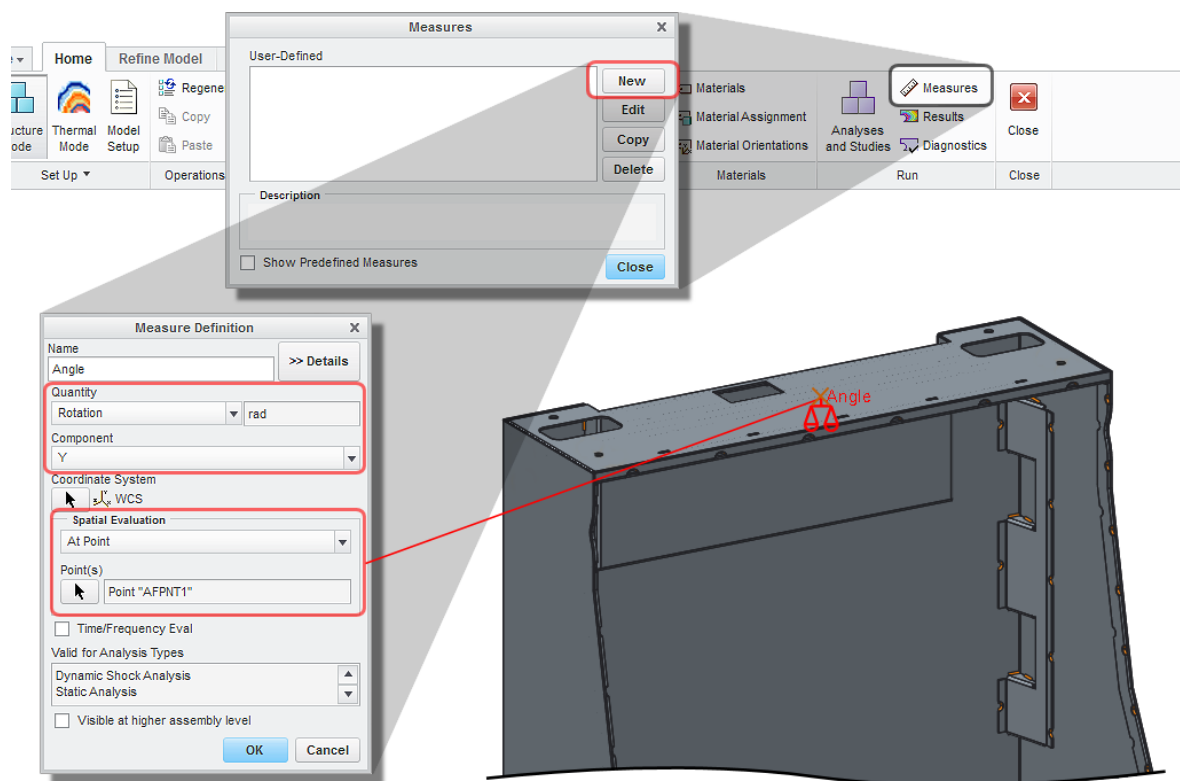


Figure 3-39: Definition of an angle measure

After the “Measure” is defined convergence method and settings will be defined in the settings of the static analysis on a separate tab (See Figure 3-40). The monitored measure is picked and a convergence criterion of 10% is set. 10% accuracy is generally enough to get a significant result in present simulation.

Due to rather coarse settings for mesh creation (even with the recommended settings for shell elements), element size is on the bigger side. Thus, in many of the analyzed cases element order needed to raise up to 8 or 9 in order to obtain convergence. As a consequence limits for polynomial order should be set to 3 minimum and 9 maximum.

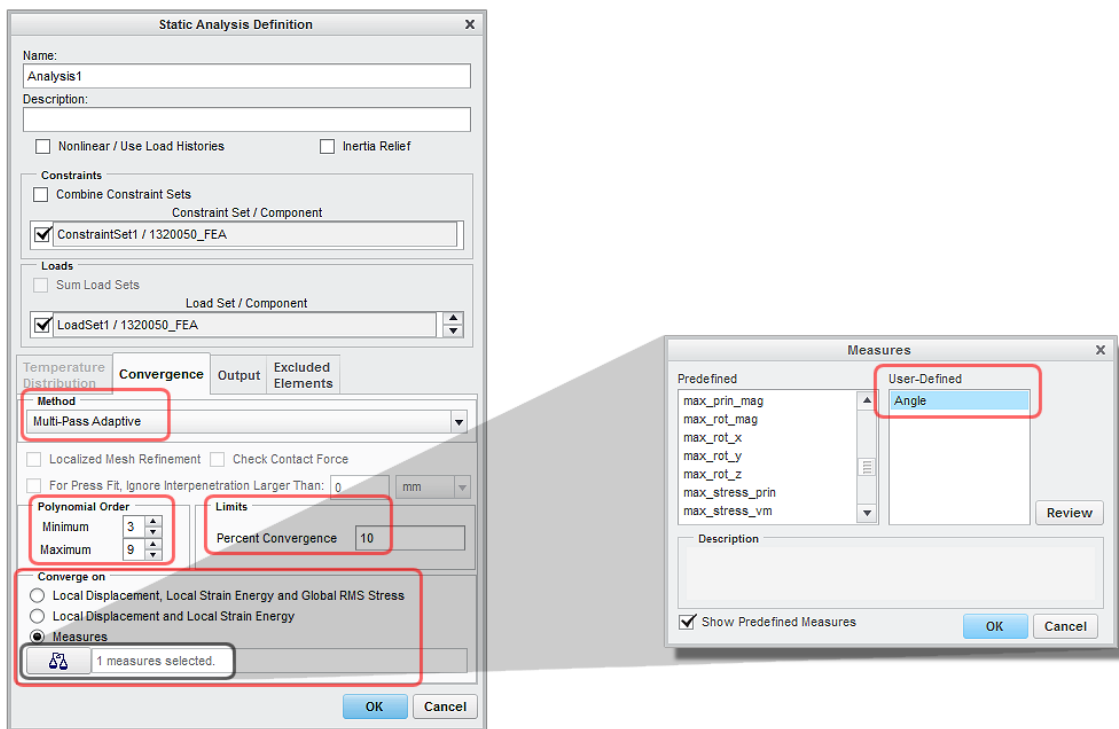


Figure 3-40: Setup for convergence criteria

In the following, Figure 3-41 shows a comparison of results for rotational deformation of different methods of convergence and their results of the individual passes. This figure shows that accuracy of the faster SPA method is very close to the MPA 1%, which represents a result that is maximum 1% off of the exact solution. In the shown calculation the MPA 10% run is farther off than the SPA. This relation between the results is representative for all analysis conducted in the course of this thesis.

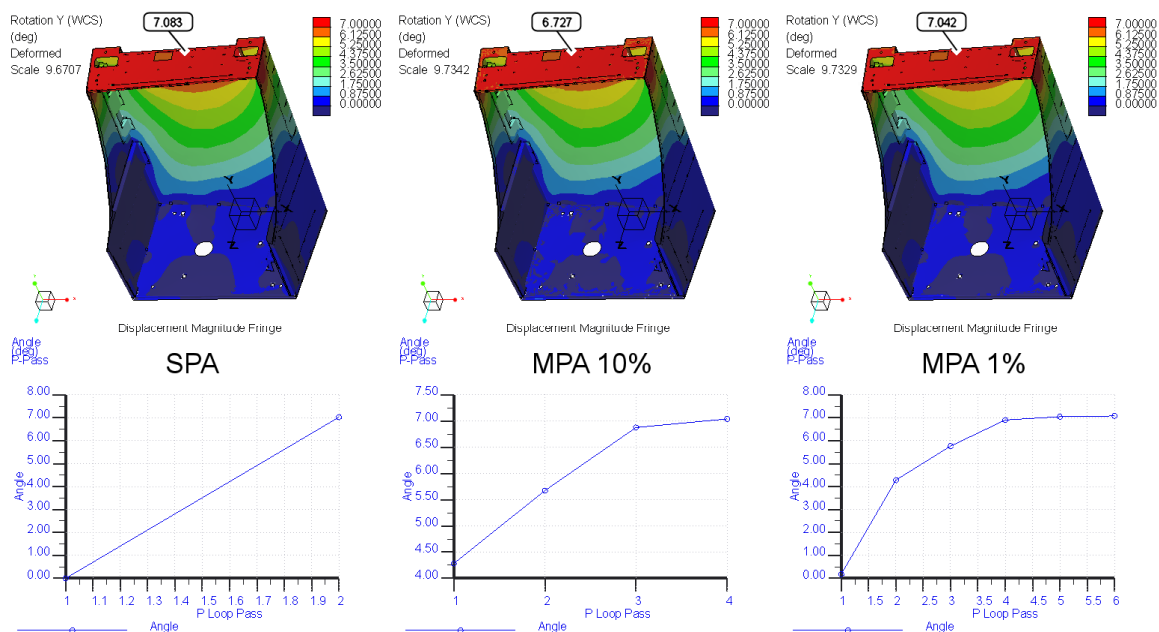


Figure 3-41: Results of convergence passes of different methods

It was shown that a result from the SPA analysis is closer to the solution than a MPA with 10% allowed deviation. Now Figure 3-42 shows the elapsed time needed for the calculation of the different methods. It's clearly evident that SPA is far more effective in giving an accurate result in short time than the MPA method. On the other side, the disadvantage is that it doesn't provide any information on how far off the result is from the solution at a max. But it can be assumed it's more exact than a 10% MPA.

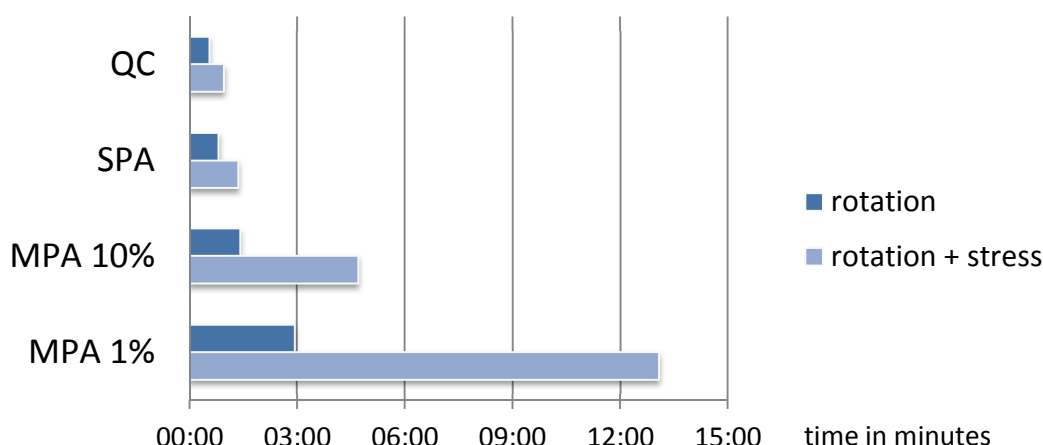


Figure 3-42: Comparison calculation time of convergence methods

As a consequence, the best option to be used generally is the SPA method. However, MPA should be used at least once for a newly generated model or mesh to get certainty of the result and check if the number of elements is sufficient to deliver a reasonable result with a polynomial order up to 9.

3.3 Post Processing

Greatest value in examining the results can only be achieved with proper display or post processing of the calculated solution. The most revealing result windows in order to show the outcome of the analysis and provide guidance for the design is a displacement fringe and an animation of the deformation with a stress fringe.

The stress fringe can only be shown if stresses are enabled in the analysis definition (according to Figure 3-35). It is able to provide additional information and point out structural weak points of the design and thus areas where plastic deformation can occur as a matter of the load.

Anyhow, stress evaluation is not priority in the scope of this thesis, because special caution in setting up the model and evaluation of the result needs to be exercised. Especially max values can't be trusted, because of singularities that can occur due to point constraints and sharp edged geometry. Therefore the stress animation is only able to provide a rough impression of resulting stress in the model.

In the course of this thesis, templates for result windows were created in order to show results for rotation and displacement. These are stored with the names:

- GTECH_rotation.rwt
- GTECH_bending.rwt

Figure 3-43 and Figure 3-44 show examples of presentations of these windows.

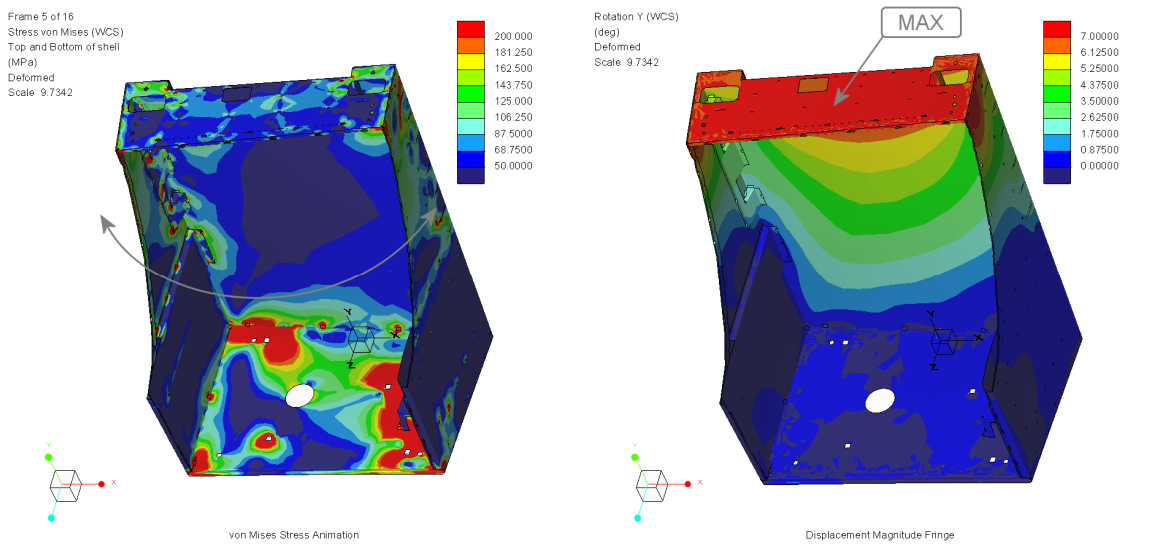


Figure 3-43: Result windows for rotation

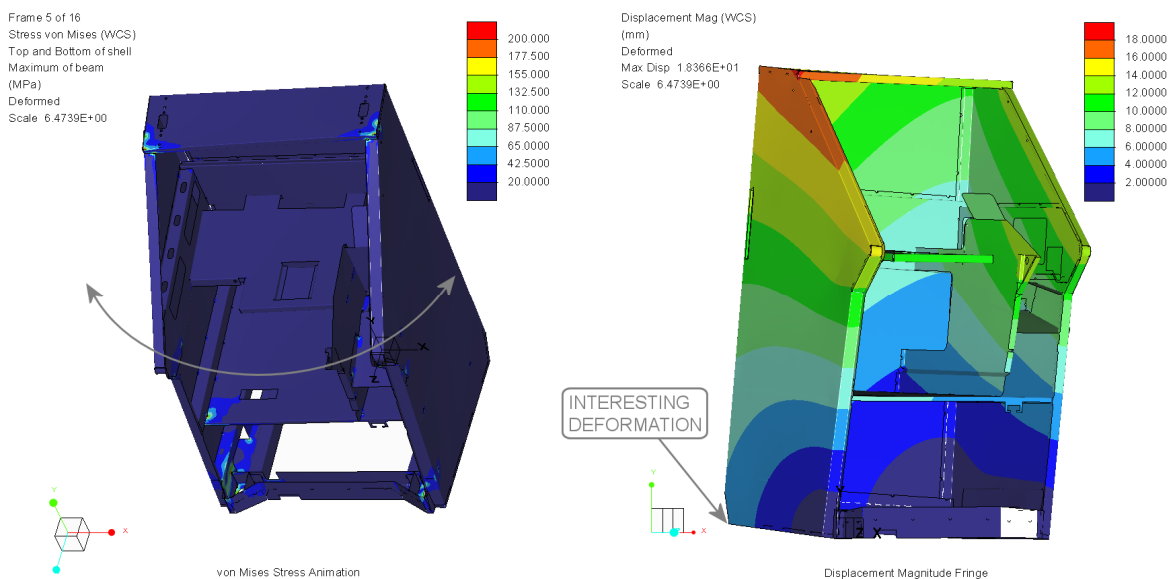


Figure 3-44: Result windows for displacement

Detailed settings of these windows are shown in the following figures. Figure 3-45 shows settings for a fringe display of displacement and rotation with the units “mm” and “degree”. Displacement and rotation is only shown in Y direction or about Y axis. Both incorporate deformation with 10% scaling.

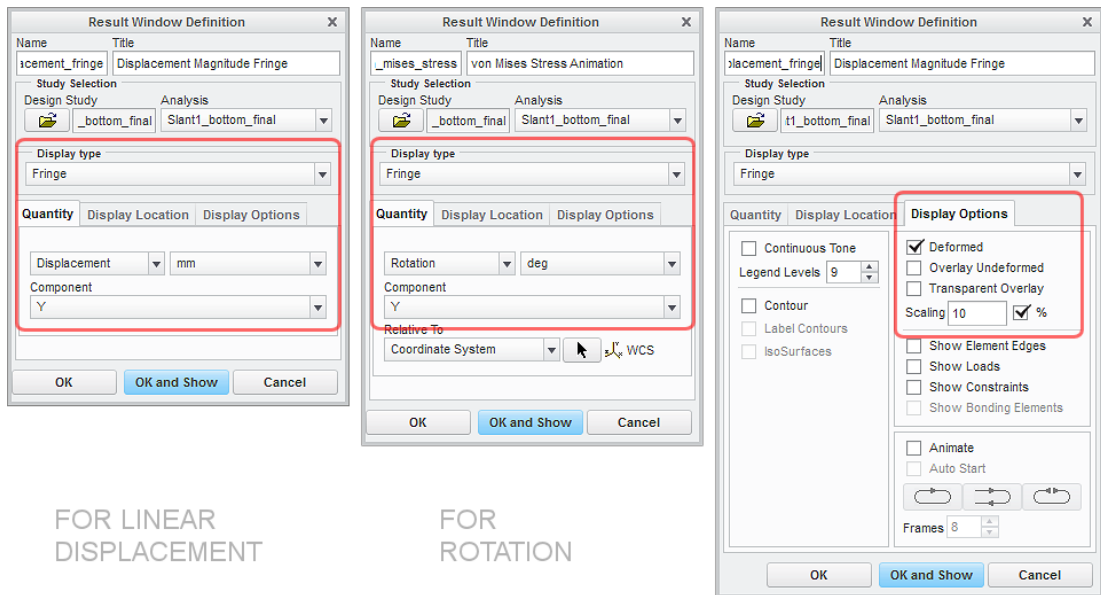


Figure 3-45: Settings for result windows – displacement and rotation

Figure 3-46 shows settings for the animated stress display with Von Mises stress and deformation in both, positive and negative directions.

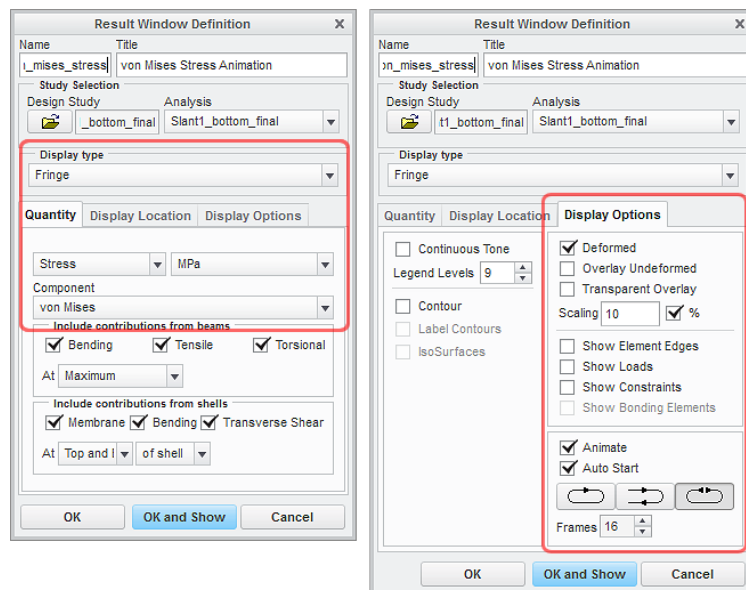


Figure 3-46: Settings for result windows – stress

4 Results

In the first section of this chapter limitations of the elaborated simulation procedure will be explained and important questions are raised that need to be verified during the evaluation process.

In the second part, results of existing cabinets with defined load cases will be presented.

4.1 Limitations

Maybe the most fundamental limitation or assumption of Creo (without the advanced extension) is something that needs to be minded generally. Materials and therefore elements are assumed to be linear and with unlimited strength. For steel, linear material behavior is generally a valid approach as long as stress doesn't exceed the yield strength of the material. This provides an essential condition that need to be considered during evaluation.

Another limiting condition, following the steps of this thesis, is that the individual components of the welded assembly are allowed to penetrate each other on places, where they would actually push against each other. As defined, the only connections between parts are typically weld connections. Therefore, parts can only transfer a load through these connections. Usually this is a good and valid approach as big areas of parts butting up against other parts without a welded connection are rare. Anyhow, it is an effect that might have significant influence in special cases and therefore should be kept in mind and additional connections should be added if applicable.

4.2 Evaluation

Generally an FEM analysis is based on a lot of simplifications and assumptions. A small change can have significant impact on the result. Therefore execution and evaluation of an analysis needs a certain amount of knowledge and a lot of experience. Else a user can easily be misled into blind acceptance of the answer produced by the program.

Therefore, an engineer should not forget his intuition and once a result of an analysis is examined the first time, the most important thing is to verify the following questions:

- Is the deformation consistent with the expectations?
- Are constraints, loads and connections working the way it's intended?

These conditions are easiest reviewed in examining an animation of the deforming process. It is likely that the result reveals an error in the setup or a missing feature in regards

to loads, constraints or connections in the course of the first run of the analysis. Thus this verification should be done carefully.

4.3 Results of Existing Cabinets

The following results of existing products present a benchmark and guidelines for future cabinet designs.

4.3.1 Oxygen Slant Top Cabinet

Generally all cabinet types should be analyzed for torsion resulting from force acting on open doors. The Oxygen Slant Top is a standard freestanding cabinet with a top-lifting door. Constraints and a load of 500N are set according to defined procedure. Figure 4-47 shows the setup on the left and the result on the right.

On the current product a rotational deformation of 5.85° is simulated. But additional investigations regarding potential improvements showed that small modifications to existing parts can bring down the deformation to around 2° .

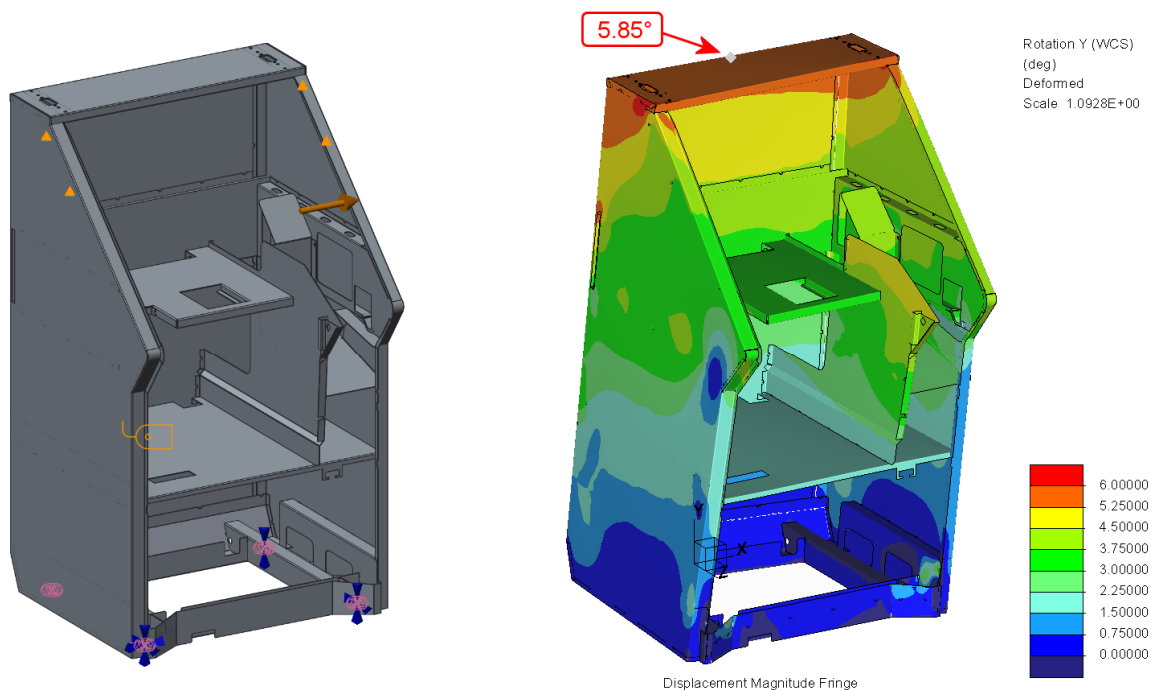


Figure 4-47: Top rotation – Oxygen Slant Top

As the Oxygen Slant Top cabinet is a freestanding cabinet it should also be tested against deformation resulting from possibly uneven floors. Therefore Figure 4-49 shows setup and result of this evaluation. Resulting deformation due to a load of 500N is 4.85mm.

Also this could be brought down significantly by incorporating small design modifications.

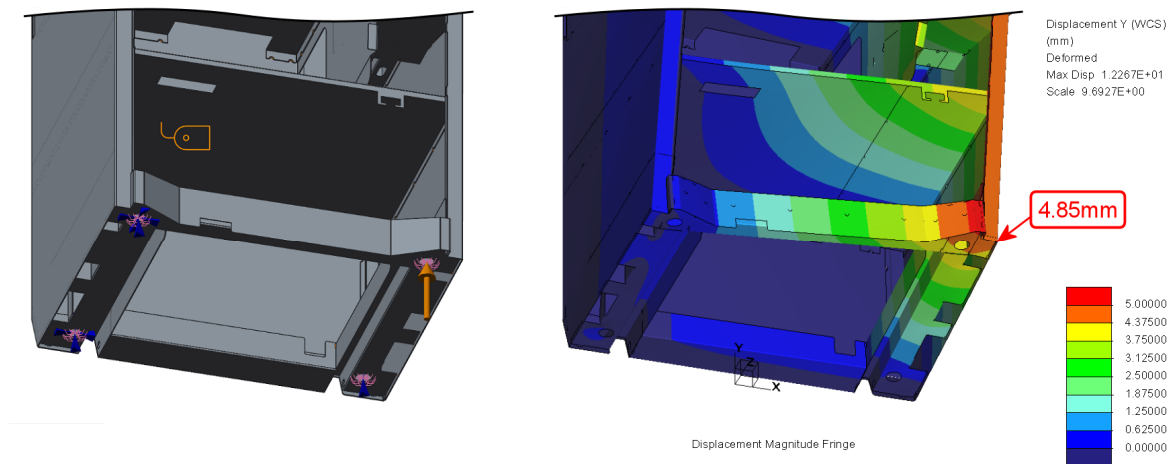


Figure 4-48: Leg displacement – Oxygen Slant Top

4.3.2 Axxis Cabinet

The Axxis is a cabinet mounted to a base with a top-lifting door. Therefore special procedure to implement constraints was followed. Figure 4-49 shows setup of constraints and load and a resulting deformation of 4.32° at the top of the cabinet.

The deformation seems to be quite high compared to the following base mounted cabinet. But this is a matter of the tight space constraints in this design and very little room for reinforcement structure on the sides.

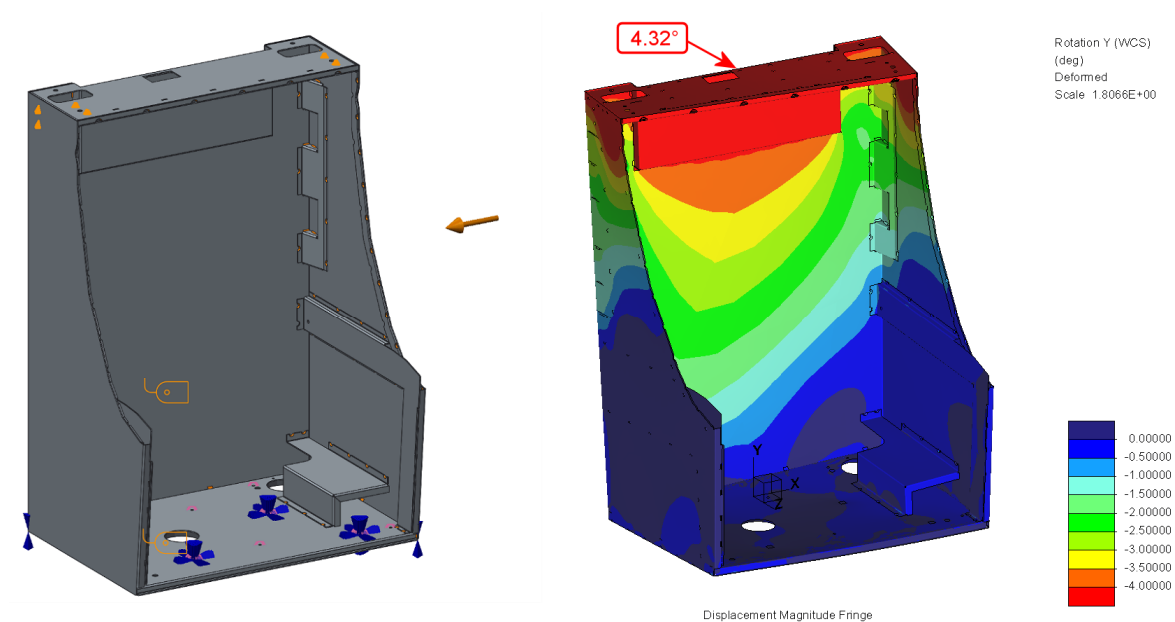


Figure 4-49: Top rotation – Axxis

4.3.3 Oxygen Upright Cabinet

Like the Axxis cabinet, the Oxygen Upright is mounted to a base. But the door opens to the side rather than to the top. Base mounting procedure and a vertical load of 500N was applied. Figure 4-50 shows setup of constraints and load and a resulting rotation of 0.42° at the top of the cabinet.

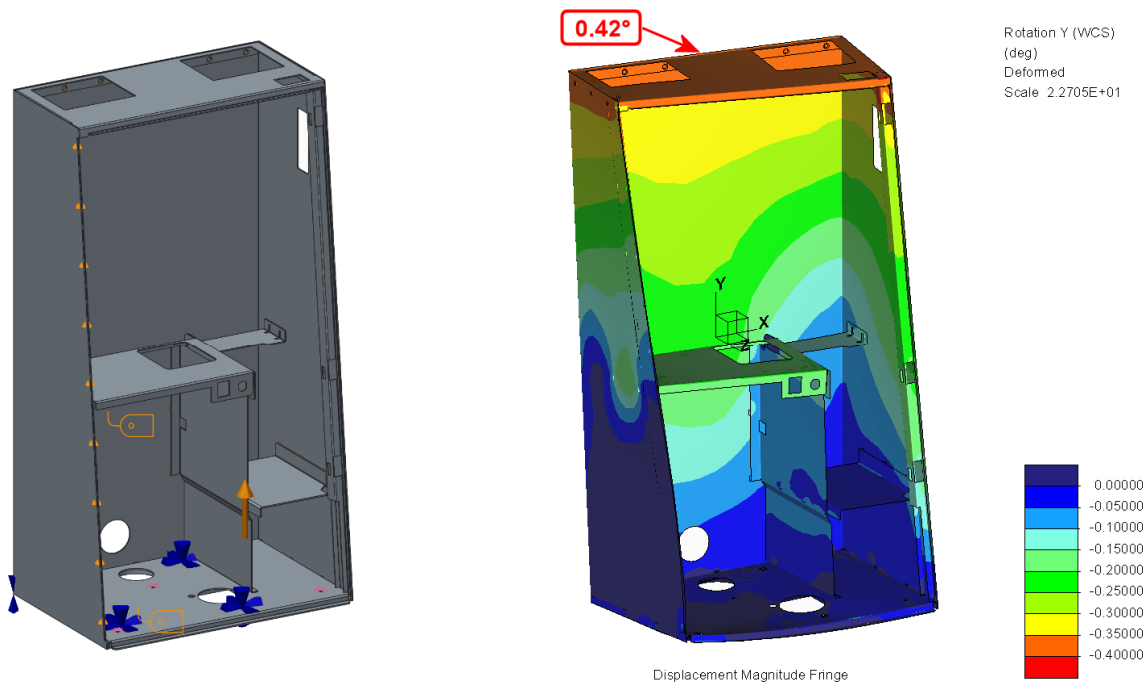


Figure 4-50: Top rotation – Oxygen

4.3.4 Stenmark Cabinet

The Stenmark cabinet is a combination of base mounted and freestanding as the actual cabinet is mounted to a freestanding base. Therefore connections between base and cabinet were incorporated with rigid links in following the principals for base mounting. Free standing constraints set was then applied to the base.

As see in Figure 4-51 the rotation on the top due to a load of 500N is 0.76° .

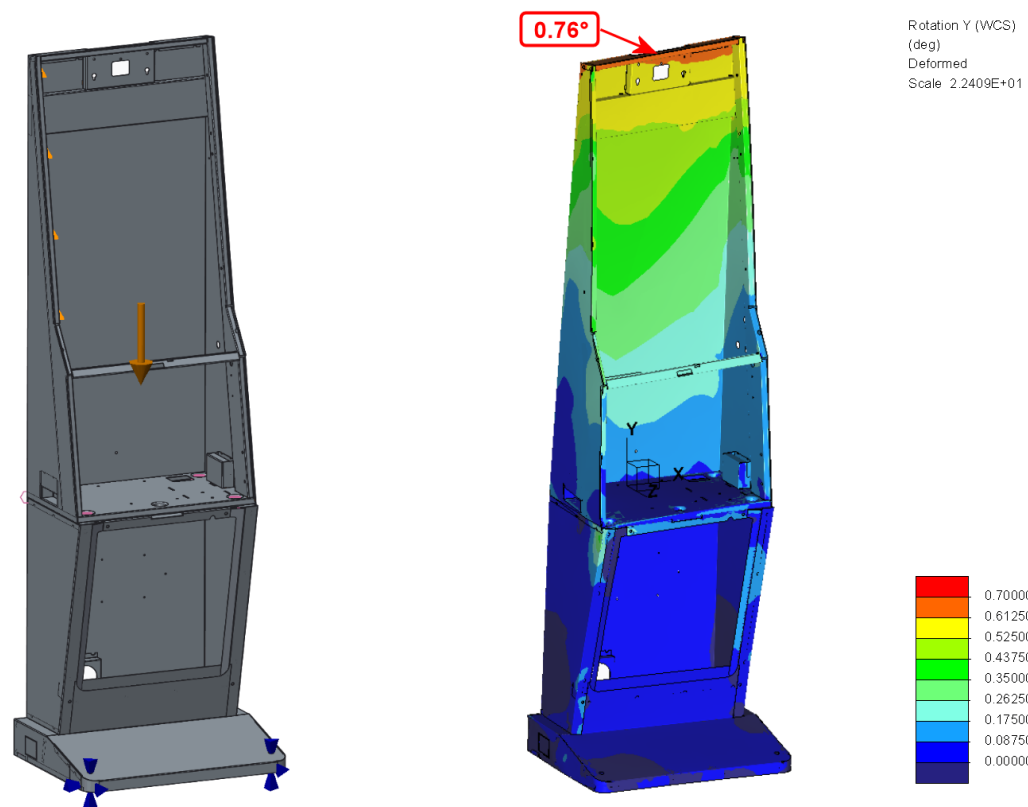


Figure 4-51: Top rotation – Stenmark

The result in Figure 4-52 is representative for the rigidity of the base plate and shows a displacement of the free leg, as a reaction to a force of 500N, of 1.42mm.

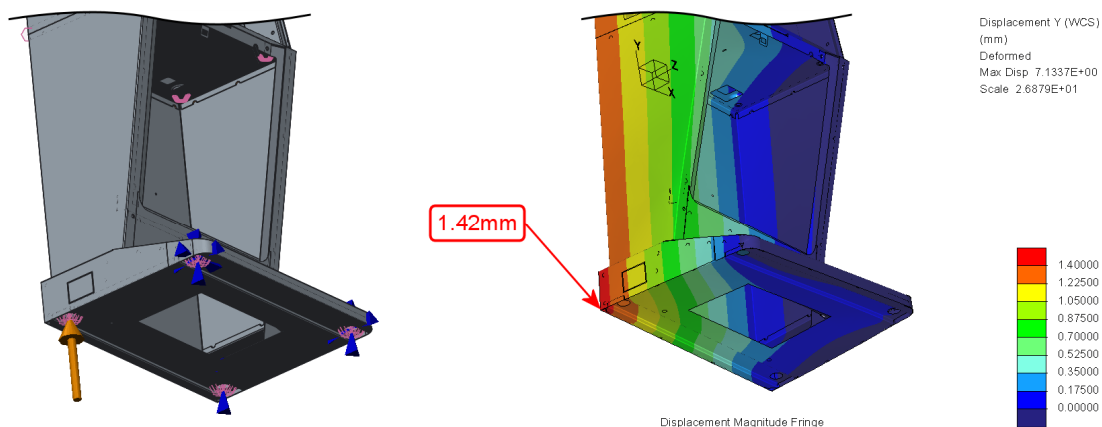


Figure 4-52: Leg displacement – Stenmark

5 Conclusion

The following sections outline the achievement of the actual project and possible further applications of FEM simulation in the area of cabinet design.

5.1 Achievement

In the course of this thesis a lot of experiments have been conducted in order to find the most effective procedures and principles to analyze welded sheet metal cabinets of gaming machines.

It was shown in general the Creo Simulate module provides all vital features necessary to include FEM simulation in the existing design process of cabinets. Important definitions already made in the CAD model like weld lines can be adopted by the simulation module quite easily. Shell features can be created almost automatically in order to replace solid sheet metal parts, which speed up the analysis drastically. Also it is very convenient revising the design and getting updated results as a fact of the integrated nature of the simulation module.

On the contrary, the “simplified representation” feature is an important feature for the purpose of allowing a continuous simulation process. Unfortunately, currently this feature is not working properly in combination with the simulation weld feature as explained in chapter 3.1.1. It is hoped that PTC is resolving this issue quickly.

A substantial part of the current paper was to create principals in order to standardize the simulation process and define reasonable set of constraints and load cases for specific cabinets and installation methods. Especially the constraint sets defined provide an important foundation for many different cases of analyses that can be performed on certain types of cabinets. In following these principals, setup of analyses can be conducted in a quick matter.

Additional investigations relating the design of existing products (results not presented in this thesis) indicated a great potential to improve the structure of cabinet designs in using simulation methods in the design process. Tests performed on some products showed that small tweaks of the structure can improve rigidity by factors of 2-3 without adding any additional parts. Furthermore, a simulation can show if parts or weld lines are redundant to some degree and thus can be considered to be removed or reduced.

Generally it can be said, in following the instructions provided by this thesis, a quick process is presented to simplify and standardize the simulation of gaming machine cabinets.

5.2 Further Applications

In the first place, further applications of the simulation module are additional or modified load cases for certain cabinet designs or cabinet features. Newly designed products (same as existing) will have different requirements and functions that can benefit from special analysis that can't be determined generally.

One of the most beneficial use cases will be optimization of designs, which are already prototyped and are showing a structural issue in a certain situation. Rigidity or stress analyses in special cases like that can be purposefully targeted on a specific problem and corrective measures can be evaluated for their effectiveness directly before another prototype is going to be built.

Further FEM simulation is also well used for parts other than sheet metal. Cast or injected parts can be analyzed against sufficient rigidity and most effective material usage. Design considerations in regards to different materials and general shape and quantity and positions of ribs can be determined on the base of simulation results.

Even though it's not checked for its practicability, rigidity and stress analyses of linkage systems will make sense. Some products have complicated hinges with up to 6 joints and very tight space constraints. This creates a challenge for the integration of enough rigidity and strength in the design. FEM simulation could help in order to optimize the design and help with design considerations.

Additional application of maybe subordinate importance could be to analyze and calculate forces and stresses resulting from deformation of parts. For example if a snap lock, a spring or a sealing lip is integrated in a plastic design, forces from predefined displacements can be calculated and keeping the design below yield strength limits of the material can be ensured.

Bibliography

- [ToRo2012] Toogood, Roger: Creo Simulate Tutorial Release 1.0 & 2.0. Edmonton: SDC Publications, 2012
- [KlBe2012] Klein Bernd: FEM: Grundlagen und Anwendungen der Finite-Element-Methode im Maschinen- und Fahrzeugbau. - 9. Aufl. Kassel: Vieweg+Teubner, 2012
- [GeCh2011] Gebhardt Christof: Praxisbuch FEM mit ANSYS Workbench. München: Carl Hanser, 2011
- [ScHe2014] Scott Hendren: Creating Midsurface Shell Idealizations. URL: <http://learningexchange.ptc.com/tutorial/50/creating-midsurface-shell-idealizations>, available on 19.5.2014, 14:00
- [UrSi2014] Urs Simmler: Wellenlagerung in Creo Simulate. URL: <http://learningexchange.ptc.com/tutorial/2573/wellenlagerung-in-creo-simulate>, available on 05.4.2014, 12:00

Statement of Authorship

I hereby certify that this diploma thesis has been composed by myself, and describes my own work, unless otherwise acknowledged in the text.

All references and verbatim extracts have been quoted, and all sources of information have been specifically acknowledged.

It has not been accepted in any previous application for a degree.

Graz, July 21, 2014

Ing. Roland Heschl